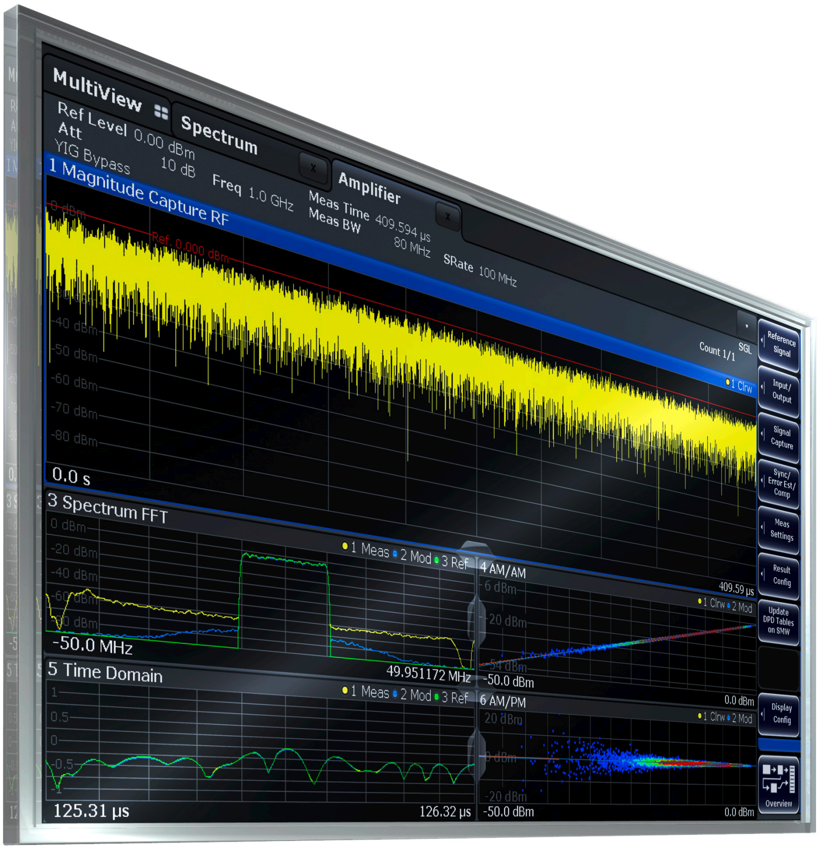
R&S®FSW-K18
Power Amplifier and Envelope Tracking Measurements
User Manual



1176.9893.02 ─ 03

Test & Measurement

User Manual

This manual describes the following R&S®FSW applications:

- R&S®FSW-K18 (1325.2170.K02)

The firmware of the instrument makes use of several valuable open source software packages. For information, see the "Open Source Acknowledgement" on the user documentation CD-ROM (included in delivery).
Rohde & Schwarz would like to thank the open source community for their valuable contribution to embedded computing.

Mühldorfstr. 15, 81671 München, Germany
Phone: +49 89 41 29 - 0
Fax: +49 89 41 29 12 164
E-mail: info@rohde-schwarz.com
Internet: www.rohde-schwarz.com

The following abbreviations are used throughout this manual: R&S®FSW-K18 is abbreviated as R&S FSW-K18.

Contents

1 Welcome to the Amplifier Measurement Application

The R&S FSW-K18 is a firmware application that adds functionality to measure the efficiency of traditional amplifiers and amplifiers that support envelope tracking with the R&S FSW signal analyzer.

This user manual contains a description of the functionality that the application provides, including remote control operation.

Functions that are not discussed in this manual are the same as in the base unit and are described in the R&S FSW User Manual. The latest versions of the manuals are available for download at the product homepage.

http://www2.rohde-schwarz.com/product/FSW.html.

Installation

Find detailed installing instructions in the Getting Started or the release notes of the R&S FSW.

1.1 Starting the Application

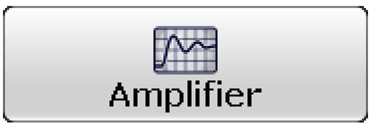
The amplifier measurement application adds a new type of measurement to the R&S FSW.

To activate the the Amplifier application

1. Press the MODE key on the front panel of the R&S FSW.

   A dialog box opens that contains all operating modes and applications currently available on your R&S FSW.

2. Select the "Amplifier" item.

   The R&S FSW opens a new measurement channel for the Amplifier application. All settings specific to amplifier measurements are in their default state.

1.2 Understanding the Display Information

The following figure shows the display as it looks for amplifier measurements. All different information areas are labeled. They are explained in more detail in the following sections.

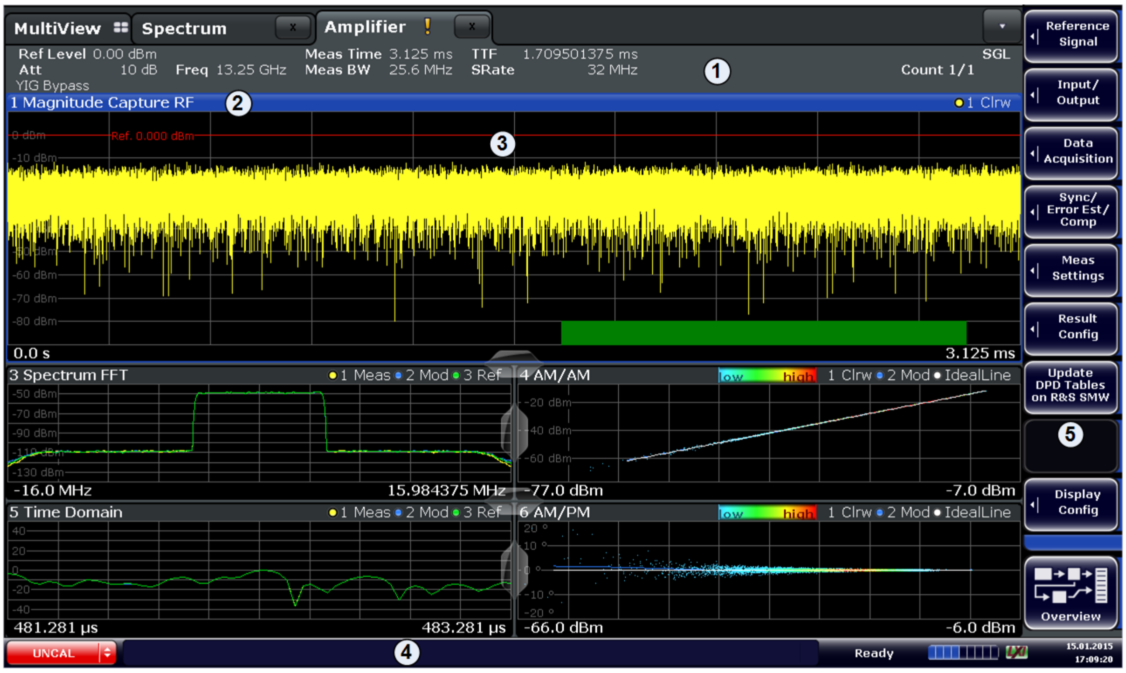


Fig. 1-1: Screen layout of the amplifier measurement application

1 = Channel bar
2 = Diagram header
3 = Result display
4 = Status bar
5 = Softkey bar

For a description of the elements not described below, please refer to the Getting Started of the R&S FSW.

Channel bar information

The channel bar contains information about the current measurement setup, progress and results.

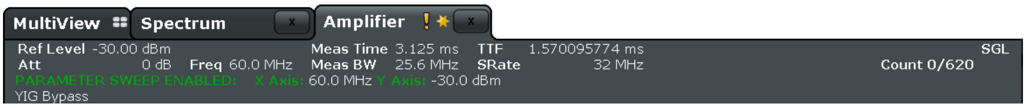


Fig. 1-2: Channel bar of the amplifier application

Ref Level	Current reference level of the analyzer.
Att	Current attenuation of the analyzer.
Freq	Frequency the signal is transmitted on.

Meas Time Length of the signal capture.

Meas BW Bandwidth with which the signal is recorded.

TTF Time difference between the trigger event and the first sample of the reference signal (= beggining of a frame).

SRate Sample rate with which the signal is recorded.

SGL Indicates that single sweep mode is active.

Count The current signal count for measurement tasks that involve a specific number of subsequent sweeps (for example the Parameter Sweep).

X Axis X-axis value that is currently measured.

Y Axis Y-axis value that is currently measured.

Window title bar information

For each diagram, the header provides the following information:

Fig. 1-3: Window title bar information of the amplifier application

1 = Window number
2 = Window type
3 = Trace color and number
4 = Trace mode

Status bar information

Global instrument settings, the instrument status and any irregularities are indicated in the status bar beneath the diagram. Furthermore, the progress of the current operation is displayed in the status bar.

2 Performing Amplifier Measurements

Note that you can use the R&S FSW-K18 with the Sequencer available with the R&S FSW. The functionality is the same as in the Spectrum application. Please refer to the R&S FSW User Manual for more information.

Numeric Result Summary

The Result Summary shows various measurement results in numerical form, combined in one table.

The table is split in two parts (three parts when you use the baseband input).

- The first part shows the modulation accuracy
- The second part shows the power characteristics of the RF signal
- The third part shows the power supply characteristics of the amplifier

7 Result Summary

Mod. Acc.	Min	Current	Max	Unit
Raw EVM	9.361	9.361	9.361	%
Raw Model EVM	0.183	29.161	721.665	%
Frequency Error	---	2.387	---	Hz
Sample Rate Error	---	0.765	---	ppm
Magnitude Error	---	0.067	---	%
Phase Error	---	4.09	---	°
Quadrature Error	---	-0.08	---	°
Gain Imbalance	---	0.013	---	dB
IQ Imbalance	---	-59.597	---	dB

For each result type, several values are displayed.

- **Current**
  Value measured during the last sweep.
  In case of measurements that evaluate each captured sample, this value represents the average value over all samples captured in the last sweep.
- **Min**

In case of measurements that evaluate each captured sample, this value represents the sample with lowest value captured in the last sweep.
- **Max**
  In case of measurements that evaluate each captured sample, this value represents the sample with the highest value captured in the last sweep.

Results that evaluate each captured sample
- Raw EVM and Raw Model EVM
- Power In and Power Out
- Gain
- All baseband results, except the Average PAE

Note: When synchronization has failed or has been turned off, some results may be unavailable.

Remote command:
Selecting the result display: LAY:ADD ? '1',LEFT,RTAB
Querying results: see chapter 5.5.3, "Retrieving Numeric Results", on page 103

Results to check modulation accuracy ← Numeric Result Summary

Raw EVM
Error vector magnitude between synchronized reference and measurement signal.
`FETCh:MACCuracy:REVM:CURRent[:RESult]?` on page 108

Raw Model EVM
Error vector magnitude between synchronized reference and model signal.
`FETCh:MACCuracy:RMEV:CURRent[:RESult]?` on page 108

Frequency Error
Difference of the RF frequency of the reference signal compared to the measured signal.

If the reference frequencies are coupled, the frequency offset should be about 0 Hz.

If the offset is very high, it is likely that the reference frequency sources are not coupled correctly, e.g. if the analyzer is configured for external reference frequency, but the cable is not connected.
`FETCh:MACCuracy:FERRor:CURRent[:RESult]?` on page 105

Sample Rate Error
Sample rate difference between reference and measurement signal.
`FETCh:MACCuracy:SRERror:CURRent[:RESult]?` on page 109

Magnitude Error
Difference in magnitude between the reference signal and the measured signal.
`FETCh:MACCuracy:MERRor:CURRent[:RESult]?` on page 107

Phase Error
Phase difference between reference and measurement signal.

If you are using the RF path for measurements, the phase between reference and measurement signal is random because the RF phases between signal generator and analyzer are not locked, even if the reference frequencies are locked. This is a typical behavior of two RF measurement instruments.
`FETCh:MACCuracy:PERRor:CURRent[:RESult]?` on page 107

Quadrature Error
Phase deviation of the 90° phase difference between the real (I) and imaginary (Q) part of the signal.

Within a typical transmitter, the I and Q signal parts are mixed with an angle of 90° by the IQ output mixer. Due to hardware imperfections, the signal delay of I and Q may be different and thus lead to an angle non-equal to 90°.
`FETCh:MACCuracy:QERRor:CURRent[:RESult]?` on page 107

Gain Imbalance
Gain difference between the real (I) and imaginary (Q) part of the signal.

This effect is typically generated by two separate amplifiers in the I and Q path of the analog baseband signal generation which have different gains.

`FETCh:MACCuracy:GIMBalance:CURRent[:RESult]?` on page 106

I/Q Imbalance
Combination of Quadrature error and Gain imbalance.

The I/Q imbalance parameter is another representation of the combination of Quadrature error and gain imbalance.

`FETCh:MACCuracy:IQIMbalance:CURRent[:RESult]?` on page 106

I/Q Offset
Shift of the measured signal compared to the ideal I/Q constellation in the I/Q plane.

`FETCh:MACCuracy:IQOFfset:CURRent[:RESult]?` on page 106

Results to check power characteristcs ← Numeric Result Summary

Power In
Signal power at the DUT input.

Should correspond to the generator output level.

`FETCh:POWer:INPut:CURRent[:RESult]?` on page 111

Power Out
Signal power at the DUT output.

`FETCh:POWer:OUTPut:CURRent[:RESult]?` on page 111

Gain
Gain of the DUT.

`FETCh:POWer:GAIN:CURRent[:RESult]?` on page 111

Crest Factor In
Crest factor of the signal at the DUT input. The crest factor is the ratio of the RMS and peak power.

`FETCh:POWer:CFACtor:IN:CURRent[:RESult]?` on page 110

Crest Factor Out
Crest factor of the signal at the DUT output. The crest factor is the ratio of the RMS and peak power.

`FETCh:POWer:CFACtor:OUT:CURRent[:RESult]?` on page 110

AM/AM Curve Width
Vertical spread of the samples in the AM/AM result display.

The spread is measured at the RMS level of the signal.

`FETCh:AMAM:CWIDth:CURRent[:RESult]?` on page 109

AM/PM Curve Width
Vertical spread of the samples in the AM/PM result display.

The spread is measured at the RMS level of the signal.

`FETCh:AMPM:CWIDth:CURRent[:RESult]?` on page 110

Compression Point (1 dB / 2 dB / 3 dB)
Input power where the gain deviates by 1 dB, 2 dB or 3 dB from a reference gain (see "Configuring compression point calculation" on page 62).

`FETCh:POWer:P1DB:CURRent[:RESult]?` on page 112

`FETCh:POWer:P2DB:CURRent[:RESult]?` on page 112

`FETCh:POWer:P3DB:CURRent[:RESult]?` on page 112

Results to check the power supply characteristics of the amplifier ← Numeric Result Summary

These results are available when you turn on the baseband input.

For valid results, make sure that you have configured the measurement correctly regarding the equipment you are using (see "Configuring PAE measurements (Power Added Efficiency)" on page 60).

Baseband Input Voltage I

Voltage measured at the I channel of the analyzer baseband input.

FETCh:IVOLtage:PURE:CURRent[:RESult]? on page 114

Baseband Input Voltage Q

Voltage measured at the Q channel of the analyzer baseband input.

FETCh:QVOLtage:PURE:CURRent[:RESult]? on page 115

Voltage

Voltage measured at the Q channel of the analyzer baseband input.

This value represents the supply voltage of the power amplifier.

The result is the same as the "Baseband Input Voltage Q" when the mulitplier = 1 and the offset = 0 (see "Configuring PAE measurements (Power Added Efficiency)" on page 60)

FETCh:VCC:CURRent[:RESult]? on page 115

Current

Current measured at the I channel of the baseband input. This corresponds to the current drawn by the amplifier.

FETCh:ICC:CURRent[:RESult]? on page 114

Power

DC power measured at the baseband input. The DC power is the product of the measured voltage and current.

FETCh:BBPower:CURRent[:RESult]? on page 114

Average PAE

The average Power Added Efficiency (PAE) indicates the efficiency of the amplifier.

The PAE is the ratio of the difference between RF output and input power and the DC power:

PAE = (Output Power - Input Power) / DC power

FETCh:APAE:CURRent[:RESult]? on page 113

Adjacent Channel Power (ACP)

The ACP result display shows the power characteristics of the transmission (Tx) channel and its neighboring channel(s).

The ACP measurement in the R&S FSW-K18 is an I/Q data based measurement. Thus, its results are calculated by the same I/Q data as the rest of the results (like the EVM). Note that the supported channel bandwidth is limited by the I/Q bandwidth of the analyzer you are using.

The results are provided in numerical form in a table. The table is made up out of two parts, one part containing the characteristics of the Tx channel, the other those of the neighboring channels.

The table contains the following information.

- **Channel**
  Shows the type of channel.
- **Bandwidth**
  Shows the channel's bandwidth (→ More information).
- **Offset** (neighboring channels only)
  Shows the frequeny offset between the center frequency of the adjacent (or alternate) channel and the center frequency of the transmission channel (→ More information).
- **Power**
  Shows the power of the transmission channel, or the power of the upper / lower neighboring channel.
- **Balanced ACP**

Shows the difference between the lower and upper adjacent channel power ("Lower Channel" - "Upper Channel").

For more information on configuring the ACP measurement see chapter 3.14, "Configuring Adjacent Channel Power (ACP) Measurements", on page 63.

Remote command:
Configuration: chapter 5.6.13, "Configuring ACP Measurements", on page 163
Result query: `CALCulate<n>:MARKer<m>:FUNCtion:POWer:RESult?` on page 163

AM/AM

The AM/AM result display shows nonlinear effects of the DUT. It shows the amplitude at the DUT input against the amplitude at the DUT output.

The ideal AM/AM curve would be a straight line at 45°. However, nonlinear effects result in a measurement curve that does not follow the ideal curve. When you drive the amplifier into saturation, the curve typically flattens at high input levels.

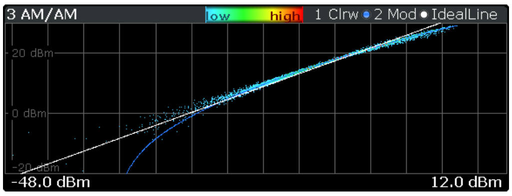
The width of the AM/AM trace is an indicator of memory effects: the larger the width of the trace, the more memory effects occur. The AM/AM Curve Width is shown in the numerical Result Summary.

Both axes show the power of the signal in dBm.

You can analyze the AM/AM characteristics of the measured signal and the modeled signal.

- Measured signal
  Shows the AM/AM characteristics of the DUT.
  The software uses the reference signal in combination with the synchronized measurement signal to calculate a software model that describes the characteristics of the device under test..
  The measured signal is represented by a colored cloud of values. The cloud is based on the recorded samples. In case of samples that have the same values (and would thus be superimposed), colors represent the statistical frequency with which a certain input / output level combination occurs in recorded samples. Blue pixels represent low statistical frequencies, red pixels high statistical frequencies. A color map is provided within the result display.
- Modeled signal
  Shows the AM/AM characteristics of the model that has been calculated. The modeled signal is calculated by applying the DUT model to the reference signal.
  When the model matches the characteristics of the DUT, the characteristics of the model signal are the same as those of the measured signal (minus the noise).
  The modeled signal is represented by a line trace.
  When system modeling has been turned off, this trace is not displayed.

All traces include the digital predistortion, when you have turned that feature on.



Remote command:
Selection: LAY:ADD AMAM
Result query: `TRACe<n>[:DATA]?` on page 102

AM/PM

The AM/PM result display shows nonlinear effects of the DUT. It shows the phase difference between DUT input and output for each sample of the synchronized measurement signal..

The ideal AM/PM curve would be a straight line at 0°. However, nonlinear effects result in a measurement curve that does not follow the ideal curve. Typically, the curve drifts from a zero phase shift, especially at high power levels when you drive the amplifier into saturation.

The width of the AM/PM trace is an indicator of memory effects: the larger the width of the trace, the more memory effects occur. The AM/PM Curve Width is shown in the numerical Result Summary.

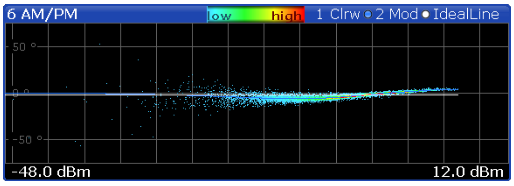
The x-axis shows the levels of all samples of the synchronized measurement signal in dBm.

The y-axis shows the phase of the signal for the corresponding power level. The unit is either rad or degree, depending on your phase unit selection in the "Display Settings".

You can analyze the AM/PM characteristics of the real DUT or of the modeled DUT.

- Measured signal
  Shows the AM/PM characteristics of the DUT.
  The software uses the reference signal together with the synchronized measurement signal to calculate a software model that describes the characteristics of the device under test.
  The measured signal is represented by a colored cloud of values. The cloud is based on the recorded samples. In case of samples that have the same values (and would thus be superimposed), colors represent the statistical frequency with which a certain input / output level combination occurs in recorded samples. A color map is provided within the result display.
- Modeled signal
  Shows the AM/PM characteristics of the model that has been calculated. The modeled signal is calculated by applying the DUT model to the reference signal.
  When the model matches the characteristics of the DUT, the characteristics of the modeled signal are the same as those of the measured signal (minus the noise).
  The modeled signal is represented by a line trace.
  When system modeling has been turned off, this trace is not displayed.

All traces include the digital predistortion, when you have turned that feature on.



Remote command:
Selection: LAY:ADD AMPM
Result query: `TRACe<n>[:DATA]?` on page 102

Gain Compression

The Gain Compression result display shows the gain and error effects of the DUT against the DUT input or output power.

The gain is the ratio of the input and output power of the DUT.

The x-axis shows the levels of all samples of the synchronized measurement signal in dBm.. You can select the information displayed on the x-axis in the "Display Settings" dialog box.

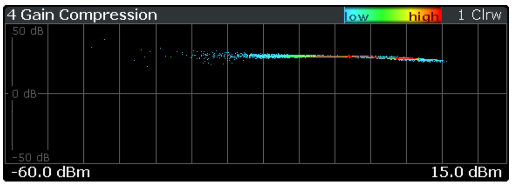
The y-axis shows the gain in dB.

The ideal Gain Compression curve would be a straight horizontal line. However, non-linear effects result in a measurement curve that does not follow the ideal curve. In addition, the curve widens at very low input levels due to noise influence.

The width of the Gain Compression trace is an indicator of memory effects: the larger the width of the trace, the more memory effects occur.

The x-axis shows the measured power levels in dBm. The y-axis shows the signal gain in dB.

The gain is represented by a colored cloud of values. The cloud is based on the recorded samples. In case of samples that have the same values (and would thus be superimposed), colors represent the statistical frequency with which a certain level / gain combination occurs in recorded samples. Blue pixels represent low statistical frequencies, red pixels high statistical frequencies. A color map is provided within the result display.



Remote command:
Selection: LAY:ADD GC
Result query: `TRACe<n>[:DATA]?` on page 102

Magnitude Capture (RF, I and Q)

The Magnitude Capture result display contains the raw data that has been recorded and thus represents the characteristics of the DUT.

It is available for the data recorded on the RF input and both baseband inputs (I and Q channels). (Note that the I and Q channel capture buffers are only available when parallel baseband capture has been turned on.)

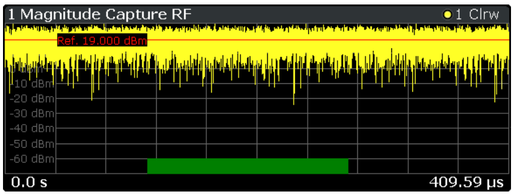
The capture buffer shows the signal level over time. The unit is either dBm (RF capture), V or A (baseband capture).

In case of the baseband capture, all multipliers and offsets are already included in the results.

The raw data is source for all further evaluations. You can also use the data in the capture buffer to identify the causes for possible unexpected results.

When you synchronize the reference signal and the measured signal, the synchronized area is indicated by a horizontal green bar on the bottom of the diagram.

The current reference level is indicated by a red horizontal line.



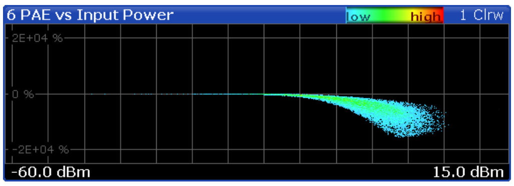
Remote command:
Selection (RF): LAY:ADD RFM
Selection (I): LAY:ADD IMAG
Selection (Q): LAY:ADD QMAG
Result query: `TRACe<n>[:DATA]?` on page 102

PAE vs Input Power / PAE vs Output Power
The PAE vs Input Power / Output Power result displays show the Power Added Efficiency (PAE) against the input or output power. It helps you to find the input or output levels at which the DUT works most efficiently.

The x-axis shows the levels of all samples of the synchronized measurement signal in dBm. The y-axis shows the efficiency in %, based on the following formula:

PAE = (RF Output Power - RF Input Power) / DC Power

The measured signal is represented by a colored cloud of values. The cloud is based on the recorded samples. In case of samples that have the same values (and would thus be superimposed), colors represent the statistical frequency with which a certain input / output level combination occurs in the recorded samples. Blue pixels represent low statistical frequencies, red pixels high statistical frequencies. A color map is provided within the result display.

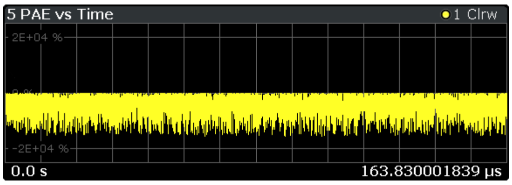


Remote command:
Selection: LAY:ADD PAEI
Result query: `TRACe<n>[:DATA]?` on page 102

PAE vs Time
The PAE Time result display shows the Power Added Efficiency against time.

The x-axis represents the time in seconds. The y-axis shows the efficiency in %, based on the following formula:

PAE = (RF Output Power - RF Input Power) / DC Power



Remote command:
Selection: LAY:OUT PAET
Result query: `TRACe<n>[:DATA]?` on page 102

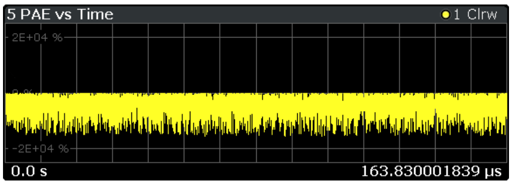
Power vs Time

The Power vs Time result display shows the supply power of the power amplifier against time.

The results are calculated by multiplying the supply voltage with the supply current which are recorded at the baseband inputs of the R&S FSW.

The unit of the results is W.

For valid results, make sure that you have configured the measurement correctly regarding the equipment you are using (see "Configuring PAE measurements (Power Added Efficiency)" on page 60).



Remote command:
Selection: LAY:ADD PVT
Result query: `TRACe<n>[:DATA]?` on page 102

Raw EVM

The Raw EVM result display shows the error vector magnitude of the signal over time.

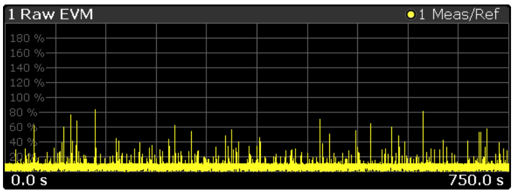
The EVM is a measure of the modulation accuracy. It compares two signals and shows the distance of the measured constellation points and the ideal constellation points.

In the R&S FSW-K18, you can compare the measured signal against the reference signal and against the modeled signal.

- Measured signal against reference signal
  Trace 1 compares the measured signal and the reference signal.
  To get useful results, the calculated linear gain is compensated to match both signals.
  Depending on the DUT, noise and nonlinear effects may have been added to the measurement signal. These effects are visualized by this trace.
- Measured signal against modeled signal
  Trace 2 compares the measured signal and the modeled signal.

The EVM between the measured and modeled signal indicates the quality of the DUT modeling. If the model matches the DUT behavior, the modeling error is zero (or is merely influenced by noise).
This result display shows changes in the model and its parameters and thus allows you to optimize the modeling.
When system modeling has been turned off, this trace is not displayed.

Note that the raw EVM is calculated for each sample that has been recorded. Thus, the raw EVM might differ from EVM values that are calculated according to a specific mobile communication standard that apply special rules to calculate the EVM, for example LTE.



Remote command:
Selection: LAY:ADD REVM
Result query: `TRACe<n>[:DATA]?` on page 102

Error Vector Spectrum

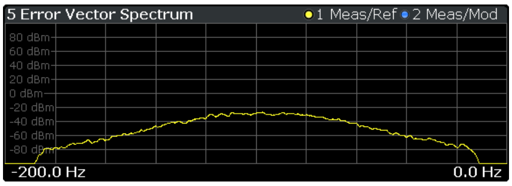
The Error Vector Spectrum result display shows the error vector (EV) signal in the spectrum around the center frequency.

The EV is a measure of the modulation accuracy. It compares two signals and shows the distance of the measured constellation points and the ideal constellation points.

The unit is dBm.

In the R&S FSW-K18, you can compare the measured signal against the reference signal and against the modeled signal.

- Measured signal against reference signal
  Trace 1 compares measured signal and the reference signal.
  To get useful results, the calculated linear gain is compensated to match both signals.
  Depending on the DUT, noise and nonlinear effects may have been added to the measurement signal. These effects are visualized by this trace.
- Measured signal against modeled signal
  Trace 2 compares measured signal and the modeled signal.
  The EVM between the measured and modeled signal indicates the quality of the DUT modeling. If the model matches the DUT behavior, the modeling error is zero (or is merely influenced by noise).
  This result display shows changes in the model and its parameters and thus allows you to optimize the modeling.
  When system modeling has been turned off, this trace is not displayed.



Remote command:
Selection: LAY:OUT SEVM
Result query: TRACe<n>[:DATA]? on page 102

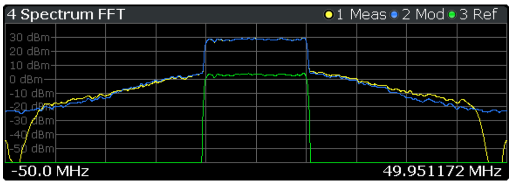
Spectrum FFT

The Spectrum FFT result display shows the frequency spectrum of the signal.

It is available for the data recorded on the RF input and both baseband inputs (I and Q channels). (Note that the Spectrum FFT of the I and Q channel are only available when parallel baseband capture has been turned on.)

The Spectrum FFT result shows the signal level in the spectrum around the center frequency. The unit is dBm.

In case of the RF spectrum, you can display the spectrum of the measured signal and the reference signal. In the best case, the measured signal has the same shape as the reference signal.



Remote command:
Selection (RF): LAY:ADD RFS
Selection (I): LAY:ADD ISP
Selection (Q): LAY:ADD QSP
Result query: TRACe<n>[:DATA]? on page 102

Time Domain

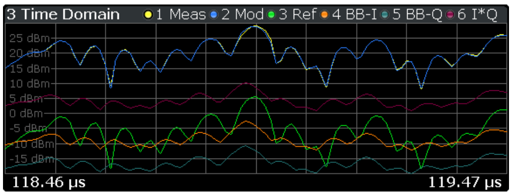
The Time Domain result display shows the signal characteristics over time.

It is similar to the Power vs Time and Magnitude Capture result displays in that it shows the signal characteristics over time. However, it deliberately shows only a very short period of the signal. You can thus use it to compare various aspects of the signal, especially the timing of the displayed signals, in a single result display.

- Measured signal
  Trace 1 shows the characteristics of the measured signal over time. The data should be the same as the results shown in the Magnitude Capture RF result display.
  In the best case, the measured signal is the same as the reference signal.
- Modeled signal
  Trace 2 shows the characteristics of the modeled signal. When system modeling has been turned off, this trace is not displayed.
  If the model matches the behavior of the DUT, the characteristics of the signal are the same as those of the measured signal (minus the noise).

- Reference signal
  Trace 3 shows the characteristics of the reference signal. The reference signal present at the DUT input represents the ideal signal.
- Current measured at the I channel of the baseband input
  Trace 4 shows the characteristics of the current that is drawn by the amplifier. It is measured at the I channel of the baseband input.
- Voltage measured at the Q channel of the baseband input
  Trace 5 shows the characteristics of the power amplifier supply voltage. It is measured at the Q channel of the baseband input.
- Power measured at the baseband input
  Trace 6 shows the power of the signal at the baseband input. The power is the product of the current and the voltage measured at the baseband channels.
  Traces 4 to 6 are available when parallel baseband capture has been turned on.

In case of the baseband capture, all multipliers and offsets are already included in the results.



Scale of the x-axis (display settings for the Time Domain)

The scale of the x-axis depends on your configuration in the "Display Settings" dialog box.

The logic is as follows:

- When you select automatic scaling (➙ "Position: Auto") and synchronization has failed, the application searches for the peak level in the capture buffer and shows the signal around the peak for the "Duration" that has been defined.
- When you select automatic scaling (➙ "Position: Auto") and synchronization is OK, the application searches for the peak level in the synchronized area of the capture buffer and shows the signal around the peak for the "Duration" that has been defined.
- When you select manual scaling (➙ "Position: Manual") and synchronization has failed, the x-axis starts at an "Offset" relative to the first sample in the capture buffer. The end of the x-axis depends on the "Duration" you have defined.
- When you select manual scaling (➙ "Position: Manual") and synchronization is OK, the x-axis starts at an "Offset" relative to the first sample in the synchronized area of the capture buffer. The end of the x-axis depends on the "Duration" you have defined.

Note: The "Display Settings" for the time domain are only available after you have selected the "Specifics for: Time Domain" item from the corresponding dropdown menu at the bottom of the dialog box.

Scale of the y-axis

The scale of the y-axis also depends on your configuration.

The signal characteristics displayed in the Time Domain result display all have a different unit. Therefore, the application provides a feature that normalizes all results to 1 (see "Configuring the Time Domain result display" on page 75). Normalization makes it easier to comapre the timing between the traces. By default, normalization is on.

Unnormalized results are displayed in their respective unit. In that case, however, the diagram might be hard to read.

Remote command:
Selection: LAY:ADD TDOM
Result query: `TRACe<n>[:DATA]?` on page 102

Vcc vs Icc
The V_{cc} vs I_{cc} result display shows the supply voltage that has been measured on baseband input Q against the current consumption that has been measured on baseband input I (using a shunt resistor or current probe).

The x-axis shows the voltage (V). The y-axis shows the current (A).

The resulting trace is usually represented by a cloud of values. The cloud is based on the recorded samples. In case of samples that have the same values (and would thus be superimposed), colors represent the statistical frequency with which a certain level / gain combination occurs in recorded samples. Blue pixels represent low statistical frequencies, red pixels high statistical frequencies. A color map is provided within the result display.

Remote command:
Selection: LAY:ADD 'VICC'
Result query: `TRACe<n>[:DATA]?` on page 102

Parameter Sweep
The Parameter Sweep result display is a result display that shows a result of the DUT (for example the EVM) against two (custom) measurement parameters. The results of this measurement are displayed in graphical and numerical form.

The Parameter Sweep is a good way to find, for example, the location of the ideal delay time of the RF signal and the envelope signal in case you are measuring an amplifier that supports envelope tracking or to determine the characteristics and behavior of an amplifier over different frequencies and levels.

For more information about supported parameters and how to set them up see "Selecting the data to be evaluated during the Parameter Sweep" on page 67.

Parameter Sweep: Diagram ← Parameter Sweep
The parameter sweep diagram is a graphical representation of the parameter sweep results. The result are either represented as a a two-dimensional trace or as a three-dimensional trace, depending on whether you are performing a parameter sweep with one or two parameters.

In a two-dimensional diagram, the y-axis always shows the result. The displayed result depends on the result type you have selected. The information displayed on the x-axis depends on the parameter you have selected for evaluation (for example the EVM over a given frequency range). Values between measurement point are interpolated. Basically, you can interpret the two-dimensional diagram as follows (example): "at a frequency of x Hz, the EVM has a value of y."

In a three-dimensional diagram, the z-axis always shows the result. The information on the other two axes is arbitrary and depends on the parameters you have selected for evaluation. For a better readability, the result values in the three-dimensional diagram are represented by a colored trace: low values have a blue color, while high values have a red color. Values between measurement point are interpolated. Basically, you can interpret the three-dimensional diagram as follows (example): "at a frequency of x Hz and a level of y, the EVM has a value of z."

Parameter Sweep: Table ← Parameter Sweep

The parameter sweep table shows the minimum and maximum results for all available result types in numerical form. For each result type, the location where the minimum and maximum result has occurred is displayed.

Example:

Result		Value	Frequency	Power
EVM	Min	0.244 %	30.0 MHz	0.0 dBm
	Max	0.246 %	10.0 MHz	0.0 dBm

A minimum EVM of 0.244 % and a maximum EVM of 0.246 % has been measured (first and second row). The minimum EVM has been measured at a frequency of 30 MHz and a output power of 0 dBm. The maximum EVM has been measured at a frequency of 10 MHz and a output power of 0 dBm.

The following result types are evaluated in the Parameter Sweep.

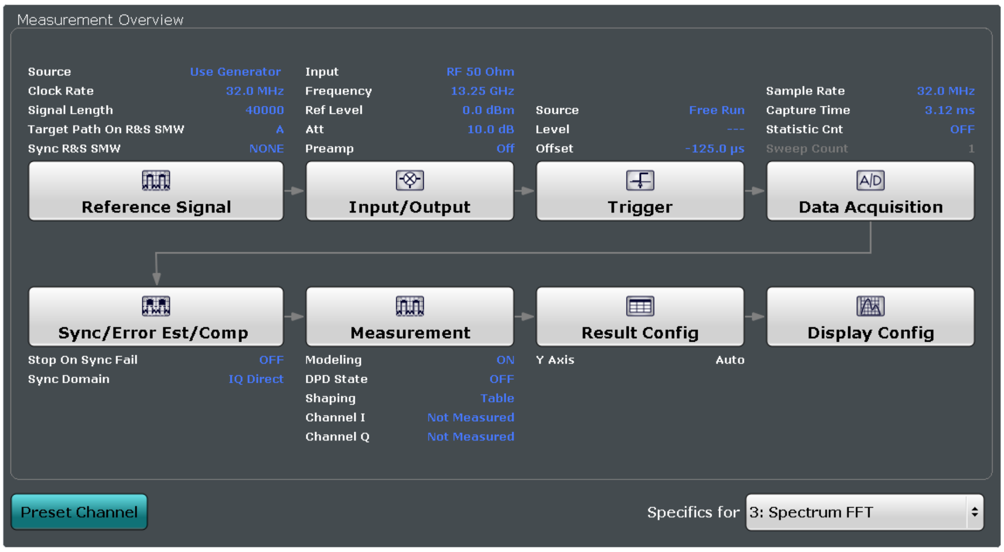
Result	Description
EVM	Error vector magnitude between synchronized reference and measurement signal.
ACP	Power of the transmission channel.
ACP Adj Upper / Lower	Power of the adjacent channels (upper and lower).
RMS Power	RMS signal power at the DUT output.
Gain	Gain of the DUT.
Crest Factor	Crest factor of the signal at the DUT output. The crest factor is the ratio of the RMS and peak power.
Curve Width (AM/AM, AM/PM)	Spread of the samples in the AM/AM (or AM/PM) result display compared to the ideal AM/AM (or AM/PM) curve.
Voltage (V_cc)	Amplifier supply voltage.
Current (I_cc)	Amplifier current consumption.
Power (V_cc * I_cc)	Amplifier DC power.
PAE	Power Added Efficiency.

Remote command:
chapter 5.5.3.3, "Retrieving Results of the Parameter Sweep Table", on page 116

3 Configuring Amplifier Measurements

3.1 Configuration Overview

Throughout the measurement channel configuration, an overview of the most important currently defined settings is provided in the "Overview". The "Overview" is displayed when you select the "Overview" icon, which is available at the bottom of all softkey menus.



In addition to the main measurement settings, the "Overview" provides quick access to the main settings dialog boxes. The individual configuration steps are displayed in the order of the data flow. Thus, you can easily configure an entire measurement channel from input over processing to output and analysis by stepping through the dialog boxes as indicated in the "Overview".

In particular, the "Overview" provides quick access to the following configuration dialog boxes (listed in the recommended order of processing):

1. Reference Signal
   See chapter 3.3, "Designing a Reference Signal", on page 27.
2. Input and output
   See chapter 3.4, "Configuring Inputs and Outputs", on page 34.
3. Trigger
   See chapter 3.5, "Triggering Measurements", on page 47.
4. Data Acquisition
   See chapter 3.6, "Configuring the Data Capture", on page 47.
5. Synchronisation, error estimation and compensation
   See chapter 3.7, "Synchronizing Measurement Data", on page 50.
   See chapter 3.9, "Estimating and Compensating Signal Errors", on page 54.
6. Measurement
   Modeling: see chapter 3.10, "Applying System Models", on page 55.
   DPD: see chapter 3.11, "Applying Digital Predistortion", on page 57.
   Envelope: see chapter 3.12, "Configuring Envelope Measurements", on page 59.
7. Result configuration
   See chapter 4, "Analysis", on page 69.
8. Display configuration
   See chapter 2, "Performing Amplifier Measurements", on page 10.

To configure settings

► Select any button in the "Overview" to open the corresponding dialog box.
  Select a setting in the channel bar (at the top of the measurement channel tab) to change a specific setting.

Preset Channel
Select the "Preset Channel" button in the lower lefthand corner of the "Overview" to restore all measurement settings **in the current channel** to their default values.

Note that the PRESET key restores the entire instrument to its default values and thus closes **all measurement channels** on the R&S FSW (except for the default Spectrum application channel)!

Remote command:
`SYSTem:PRESet:CHANnel[:EXECute]` on page 90

Specifics for
The measurement channel may contain several windows for different results. Thus, the settings indicated in the "Overview" and configured in the dialog boxes vary depending on the selected window.

Select an active window from the "Specifics for" selection list that is displayed in the "Overview" and in all window-specific configuration dialog boxes.

The "Overview" and dialog boxes are updated to indicate the settings for the selected window.

3.2 Performing Measurements

Continuous Sweep/RUN CONT

After triggering, starts the measurement and repeats it continuously until stopped. This is the default setting.

While the measurement is running, the "Continuous Sweep" softkey and the RUN CONT key are highlighted. The running measurement can be aborted by selecting the highlighted softkey or key again. The results are not deleted until a new measurement is started.

Note: Sequencer. If the Sequencer is active, the "Continuous Sweep" softkey only controls the sweep mode for the currently selected channel; however, the sweep mode only has an effect the next time the Sequencer activates that channel, and only for a channel-defined sequence. In this case, a channel in continuous sweep mode is swept repeatedly.
Furthermore, the RUN CONT key controls the Sequencer, not individual sweeps. RUN CONT starts the Sequencer in continuous mode.

Remote command:
`INITiate<n>:CONTinuous` on page 99

Single Sweep/ RUN SINGLE

After triggering, starts the number of sweeps set in "Sweep Count". The measurement stops after the defined number of sweeps has been performed.

While the measurement is running, the "Single Sweep" softkey and the RUN SINGLE key are highlighted. The running measurement can be aborted by selecting the highlighted softkey or key again.

Note: Sequencer. If the Sequencer is active, the "Single Sweep" softkey only controls the sweep mode for the currently selected channel; however, the sweep mode only has an effect the next time the Sequencer activates that channel, and only for a channel-defined sequence. In this case, a channel in single sweep mode is swept only once by the Sequencer.
Furthermore, the RUN SINGLE key controls the Sequencer, not individual sweeps. RUN SINGLE starts the Sequencer in single mode.

If the Sequencer is off, only the evaluation for the currently displayed measurement channel is updated.

Remote command:
`INITiate<n>[:IMMediate]` on page 99

Continue Single Sweep
While the measurement is running, the "Continue Single Sweep" softkey and the RUN SINGLE key are highlighted. The running measurement can be aborted by selecting the highlighted softkey or key again.

Remote command:
`INITiate<n>:CONMeas` on page 98

3.3 Designing a Reference Signal

Many of the results available in the R&S FSW-K18 require a reference signal that describes the characteristics of the signal you feed into the amplifier.

The reference signal describes the characteristics of the signal that you feed into the amplifier and whose amplified version is measured by the application. You can define any signal you want as a reference signal.

The application provides several methods to design a reference signal:

- Designing the signal on a generator
  (Having a Rohde & Schwarz generator is mandatory for this method.)
- Designing the signal in a waveform file
- Designing the signal in the R&S FSW-K18
  (Having a Rohde & Schwarz generator is mandatory for this method.)

For a list of supported signal generators, refer to the datasheet of the R&S FSW-K18.

► To access reference signal settings, proceed as follows:
  - In the "Configuration Overview", select the "Reference Signal" button.
  - Press the MEAS CONFIG key and then select the "Reference Signal" softkey.

The dialog box to configure the reference signal is made up out of three tabs. Each tab represents one of the available methods to design the reference signal.

Signal information

Each tab of the "Reference Signal" dialog box contains some basic information about the reference signal that is currently in use.

The information is only displayed when a reference signal has been successfully loaded. When you load a different waveform, the reference signal information is updated accordingly.

- Sample rate
  The sample rate in the header of the currently used reference signal waveform file in Hz.
- Sample length
  Length of the currently used reference signal waveform file in samples.

- Waveform file
  Name and path of the waveform file currently in use.

Remote command:

Sample rate: `CONFigure:REFSignal:SINFo:SRATe?` on page 133

Sample length: `CONFigure:REFSignal:SINFo:SLENgth?` on page 133

Using multi segment waveform files

Modern chip technologies implement several communication standards within one chip and thus increase the requirements in spatial design and test systems. To fulfill the requirements in the test systems, and to enable a rapid change between different waveforms containing different test signals, the R&S SMW provides the functionality to generate multi segment waveform files, files that contain several different waveforms.

(For more information about creating and using multi segment waveform files (including examples) refer to the documentation of the R&S SMW.)

When you are testing amplifiers with the R&S FSW-K18, you can use a multi segment waveform file to create the reference signal. If you use one of these, you have to select the segment that you want to use as a reference signal in the corresponding input field.

Note that the content of the segment you are using for the reference signal has to match the content of the segment that is currently used by the ARB of the signal generator. You can select the segment for the used by the generator in the Generator Setup.

Remote command:

`CONFigure:REFSignal:SEGMent` on page 133

Transferring the reference signal

Both the signal generator and analyzer used in the test setup need to know the characteristics of the reference signal.

- The signal generator needs that information to generate the signal.
- The analyzer needs that information for the evaluation of the results.

This is why you have to transfer the signal information to both instruments. The transmission is done through a LAN connection that you have to establish when setting up the measurement. For more information on that see chapter 3.4.6, "Controlling a Signal Generator", on page 44.

- When you design the reference signal on the signal generator, transfer the signal information from the generator to the analyzer with the ➙"Read and Load Current Signal from R&S SMW" button.
  You can either design a reference signal with one of the available firmware options (for example an LTE signal with the R&S SMW-K55) or design a signal in a custom waveform file. Note that the R&S FSW-K18 does not support all firmware options of the R&S SMW.
- When you load the reference signal from a waveform file or design the signal within the R&S FSW-K18, transfer the signal information from the analyzer to the generator. Depending on the signal source, you can do this either with the "Load and Export Selected Waveform File to R&S SMW" or the "Generate and Load Signal and Export it to R&S SMW" buttons.

When you send the signal information to the generator, the application automatically configures the generator accordingly.

Transmission state

The LED displayed with the transmission button shows the state of the reference signal transmission.

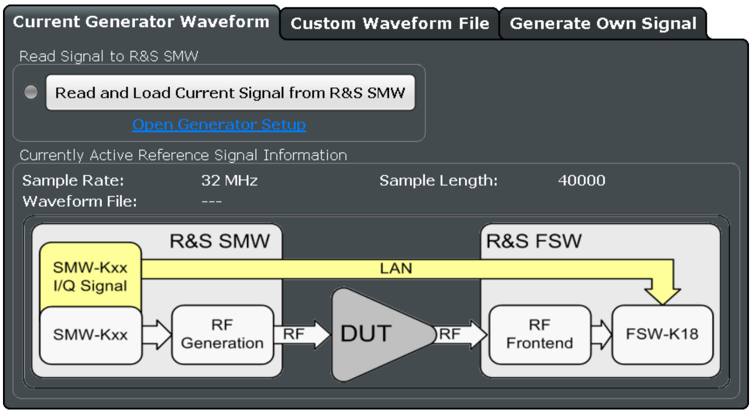
The LED is either grey, green or red:

- Grey LED
  Transmission state unknown (for example when you have not yet started the transmission).
- Green LED
  Transmission has been successful.
- Red LED
  Transmission has not been successful.
  Check if the connection between analyzer and generator has been established or if the IP address has been stated correctly.

Designing a reference signal on a signal generator

One way to design a reference signal is to design the signal on the signal generator itself.

You can design any signal you like, as long as it is storable as an arbitrary waveform (ARB) file. When you are done, you have to transfer the signal information from the signal generator to the signal analyzer with the "Read Signal from R&S SMW" button.



Most of the options available for the R&S SMW are supported by the automatic signal import functionality of the R&S FSW-K18. If the signal import was not successful (indicated by a red LED), you have to transfer the reference signal in another way (for example with a memory stick).

For a comprehensive description of all features available on the signal generator and information on how to generate signals, please refer to the documentation of the signal generator.

Remote command:
See signal generator documentation.
`CONFigure:REFSignal:CGW:READ` on page 128
`CONFigure:REFSignal:CGW:LEDState?` on page 127

Designing a reference signal in a waveform file

One way to design a reference signal is to define its characteristics in a waveform file (*.wv or *.iq.tar).

You can create a waveform file, for example

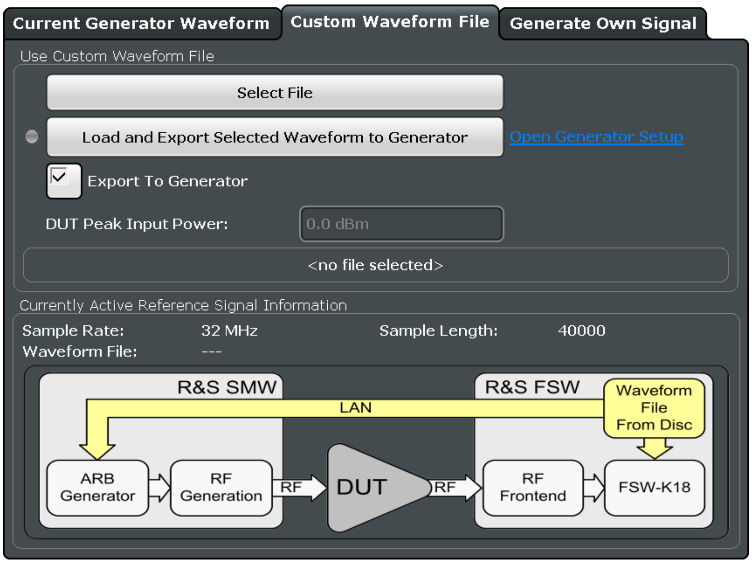
- with the R&S®WinIQSIM2 software package
- by exporting a signal designed on the signal generator

Basically, this file contains the characteristics of the reference signal. The generator then generates the reference signal based on the information in the file.

There are two ways to generate the reference signal through a custom waveform file.

- The generator is connected to the R&S FSW in a LAN, and can be recognized by the R&S FSW-K18 (Rohde & Schwarz generators only, for example the R&S SMW)
  In that case you can simply transfer the reference signal information to the generator with the features integrated into the R&S FSW-K18. This then generates the corresponding signal with the appropriate signal level, and the R&S FSW-K18 is able to compare the measured signal to the ideal reference signal.
- The generator is not connected to the R&S FSW
  In that case, you have to load the reference signal information onto the generator manually and turn off the "Export to Generator" function. Because no exchange of information is possible between generator and analyzer, it is required to specify the input level of the signal in the "DUT Peak Input Power" input field.

For a comprehensive description of all features available on the signal generator and information on how to generate and export signals to a file, please refer to the documentation of the signal generator.



To transfer a waveform file from the analyzer to the generator and process it with the ARB generator of the R&S SMW, for example, proceed as follows:

► In the "Custom Waveform" tab, select a file via the "Select File" button.

► Transfer the file to the generator with the "Load and Export Selected Waveform to generator" button.

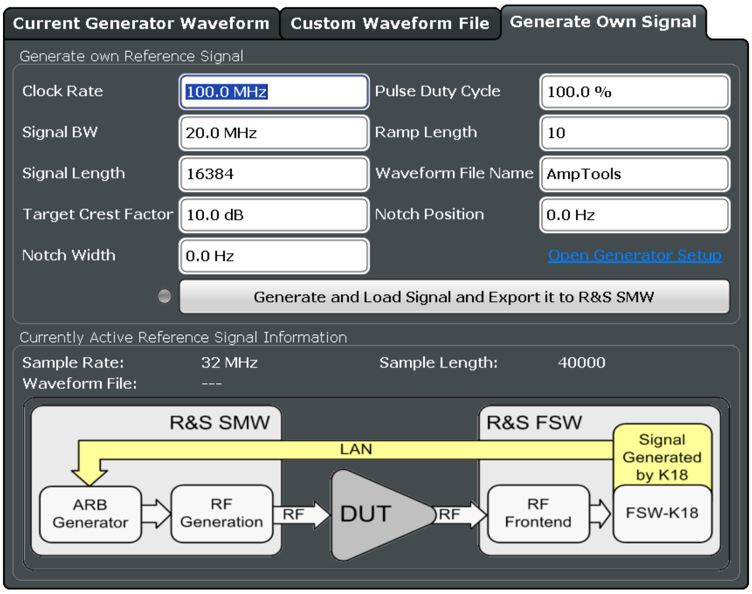
Remote command:
Select file: `CONFigure:REFSignal:CWF:FPATh` on page 129
Transfer file: `CONFigure:REFSignal:CWF:WRITe` on page 129
Transmission state: `CONFigure:REFSignal:CWF:LEDState?` on page 129
Export file: `CONFigure:REFSignal:CWF:ETGenerator[:STATe]` on page 128
DUT input power: `CONFigure:REFSignal:CWF:DPIPower` on page 128

Designing a reference signal within the R&S FSW-K18
One way to design a reference signal is to design the signal within the R&S FSW-K18.

The application provides functionality to design a basic reference signal and saves the signal characteristics in a waveform file which you have to transfer to the signal generator with the "Send Signal to R&S SMW" button.

When the data has been transfered, the signal generator (for example the R&S SMW) generates the corresponding signal.

The generated signal is a multicarrier signal with OFDM characteristics, whose basic properties, like crest factor and bandwidth, you can specify as required.



To generate a reference signal within the application, proceed as follows:

► In the "Generated Reference Signal" tab, design the reference signal as required.

The application stores the current signal properties as an ARB signal in a waveform file.

► Upload the data to the generator with the "Send Signal to R&S SMW" button.

You can define the following signal characteristics.
- "Clock Rate" on page 32
- "Signal Bandwidth" on page 32
- "Signal Length" on page 33
- "Crest Factor" on page 33
- "Notch Width" on page 33
- "Notch Position" on page 33
- "Pulse Duty Cycle" on page 33
- "Ramp Length" on page 34
- "Waveform File Name" on page 34

Remote command:
`CONFigure:REFSignal:GOS:WRITe` on page 133
`CONFigure:REFSignal:GOS:LEDState?` on page 130

Clock Rate ← Designing a reference signal within the R&S FSW-K18
Defines the clock or sample rate that the reference signal is generated with.

The purpose of the application is to measure nonlinear effects. These generate spectral regrowth (amplitude components in addition to the signal).

Remote command:
`CONFigure:REFSignal:GOS:SRATe` on page 132

Signal Bandwidth ← Designing a reference signal within the R&S FSW-K18
Defines the bandwidth of the reference signal.

The bandwidth should not be larger than maximum I/Q bandwidth supported by your signal analyzer (which depends on the analyzer configuration).

Remote command:
`CONFigure:REFSignal:GOS:BWIDth` on page 130

Signal Length ← Designing a reference signal within the R&S FSW-K18
Defines the number of samples that the reference signal consists of.

A number that is a power of 2 will speed up the internal signal processing. Thus, such a number should be specified if no other requirements limit the choice of the sample count.

For more information see "Pulse Duty Cycle" on page 33.

Remote command:
`CONFigure:REFSignal:GOS:SLENgth` on page 132

Crest Factor ← Designing a reference signal within the R&S FSW-K18
Defines the crest factor of the reference signal.

The crest factor shows the RMS power in relation to the peak power.

Remote command:
`CONFigure:REFSignal:GOS:CRESt` on page 130

Notch Width ← Designing a reference signal within the R&S FSW-K18
Defines the width of a notch that you can add to the reference signal.

Within the notch, all carriers of the reference signal have zero amplitude. You can use the noise notch to, for example, determine the noise power ratio (NPR) before and after the DPD.

Remote command:
`CONFigure:REFSignal:GOS:NWIDth` on page 131

Notch Position ← Designing a reference signal within the R&S FSW-K18
Defines an offset for the noise notch relative to the center frequency.

The offset moves the notch to a position outside the center of the signal. You can use the offset to, for example, generate a one-sided noise signal or to examine asymmetric distortion effects.

Remote command:
`CONFigure:REFSignal:GOS:NPOSition` on page 131

Pulse Duty Cycle ← Designing a reference signal within the R&S FSW-K18
Defines the duty cycle of a pulsed reference signal.

The duty cycle of a pulse is the ratio of the pulse duration and the actual length of the pulse. A duty cycle of 100 % correposnds to a continuous signal.

Example:

The pulse duration is 2 μs. The actual length of the pulse is 1 μs. The duty cycle is 1 μs : 2 μs = 0.5 or 50 %.

Remote command:
`CONFigure:REFSignal:GOS:DCYCle` on page 130

Ramp Length ← Designing a reference signal within the R&S FSW-K18
Defines the number of samples used to ramp up the pulse to its full power and vice versa.

Remote command:
`CONFigure:REFSignal:GOS:RLENgth` on page 131

Waveform File Name ← Designing a reference signal within the R&S FSW-K18
Defines the name of the waveform file that the reference ARB signal configuration is stored in.

Remote command:
`CONFigure:REFSignal:GOS:WNAMe` on page 132

3.4 Configuring Inputs and Outputs

3.4.1 Selecting and Configuring the Input Source

The R&S FSW-K18 allows you to use the RF input and the Analog Baseband input which is available with option R&S FSW-B71.

Simultanous use of the RF input and the Analog Baseband input
Compared to other applications available for the R&S FSW, the R&S FSW-K18 allows you to use both the RF input and the Analog Baseband input simultaneously.

This allows for various specific measurements which require a simultaneous capture of the RF signal, of the supply voltage and of the current drawn by an amplifier. Such a test setup is, for example, required to calculate the instantaneous PAE (Power Added Efficiency), which in turn is of interest for measurements on amplifiers that make use of envelope tracking.

You can configure the signal inputs in the "Input Source" tab of the "Input / Output" dialog box.

► To access the input source settings, proceed as follows:
   - In the "Configuration Overview", select the "Input / Output" button and then the "Input Source" tab.

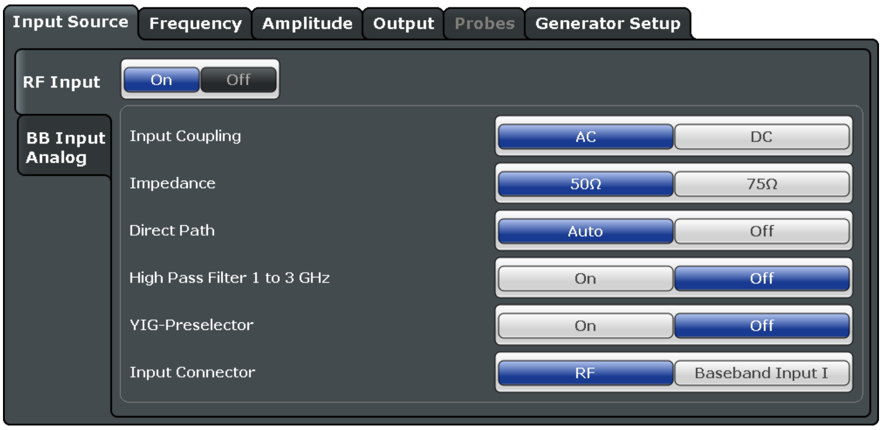
- Press the INPUT / OUTPUT key and then select the "Input Source Config" softkey.

3.4.1.1 Configuring the RF Input

The RF input captures the RF signal that you are measuring. It is always on.

The RF input source characteristics are similar to those available in the Spectrum application.For a comprehensive description of these settings, please refer to the R&S FSW User Manual.

► Select the "RF Input" tab (vertical) from the "Input Source" dialog box.



Input Coupling
The RF input of the R&S FSW can be coupled by alternating current (AC) or direct current (DC).

AC coupling blocks any DC voltage from the input signal. This is the default setting to prevent damage to the instrument. Very low frequencies in the input signal may be distorted.

However, some specifications require DC coupling. In this case, you must protect the instrument from damaging DC input voltages manually. For details, refer to the data sheet.

Remote command:
`INPut:COUPling` on page 135

Impedance
For some measurements, the reference impedance for the measured levels of the R&S FSW can be set to 50 Ω or 75 Ω.

75 Ω should be selected if the 50 Ω input impedance is transformed to a higher impedance using a 75 Ω adapter of the RAZ type (= 25 Ω in series to the input impedance of the instrument). The correction value in this case is 1.76 dB = 10 log (75Ω/50Ω).

Remote command:
`INPut:IMPedance` on page 136

Direct Path
Enables or disables the use of the direct path for small frequencies.

In spectrum analyzers, passive analog mixers are used for the first conversion of the input signal. In such mixers, the LO signal is coupled into the IF path due to its limited isolation. The coupled LO signal becomes visible at the RF frequency 0 Hz. This effect is referred to as LO feedthrough.

To avoid the LO feedthrough the spectrum analyzer provides an alternative signal path to the A/D converter, referred to as the *direct path*. By default, the direct path is selected automatically for RF frequencies close to zero. However, this behavior can be deactivated. If "Direct Path" is set to "Off", the spectrum analyzer always uses the analog mixer path.

"Auto" (Default) The direct path is used automatically for frequencies close to zero.

"Off" The analog mixer path is always used.

Remote command:
`INPut:DPATh` on page 135

High-Pass Filter 1...3 GHz
Activates an additional internal high-pass filter for RF input signals from 1 GHz to 3 GHz. This filter is used to remove the harmonics of the analyzer in order to measure the harmonics for a DUT, for example.

This function requires an additional hardware option.

(Note: for RF input signals outside the specified range, the high-pass filter has no effect. For signals with a frequency of approximately 4 GHz upwards, the harmonics are suppressed sufficiently by the YIG filter.)

Remote command:
`INPut:FILTer:HPASs[:STATe]` on page 135

YIG-Preselector
Activates or deactivates the YIG-preselector, if available on the R&S FSW.

An internal YIG-preselector at the input of the R&S FSW ensures that image frequencies are rejected. However, this is only possible for a restricted bandwidth. In order to use the maximum bandwidth for signal analysis you can deactivate the YIG-preselector at the input of the R&S FSW, which may lead to image-frequency display.

Note that the YIG-preselector is active only on frequencies greater than 8 GHz. Therefore, switching the YIG-preselector on or off has no effect if the frequency is below that value.

Remote command:
`INPut:FILTer:YIG[:STATe]` on page 136

Input Connector
Determines whether the RF input data is taken from the RF INPUT connector (default) or the optional BASEBAND INPUT I connector. This setting is only available if the optional Analog Baseband Interface is installed and active for input. It is not available for the R&S FSW67 or R&S FSW85.

This feature is available when you turn off Enable Parallel BB Capture.

For more information on the Analog Baseband Interface (R&S FSW-B71) see the R&S FSW I/Q Analyzer and I/Q Input User Manual.

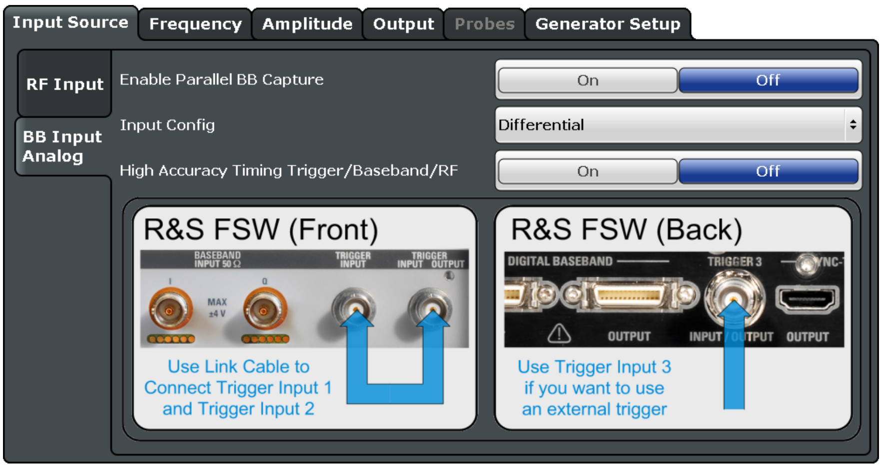
Remote command:
`INPut:CONNector` on page 134

3.4.1.2 Configuring the Analog Baseband Input

The analog baseband input input is available as a hardware option.

For measurements that also take into account the supply voltage and the current drawn by the PA, the analog baseband inputs are required to measure the voltage (baseband input Q) and the current (baseband input I). Typically some power probes have to be connected to the baseband inputs for this purpose.

► Select the "BB Input Analog" tab (vertical) from the "Input Source" dialog box.



Enable Parallel BB Capture
Turns simultaneous data capture on the RF input and the analog baseband input on and off.

This is necessary when you perform measurements that take into account the supply voltage and the current drawn by the PA.

Remote command:
`INPut:SELect:BBANalog[:STATe]` on page 137

Input Configuration
Defines whether the input is provided as a differential signal via all four Analog Baseband connectors or as a plain I/Q signal via two simple-ended lines.

Note: Both single-ended and differential probes are supported as input; however, since only one connector is occupied by a probe, the "Single-ended" setting must be used for all probes.

"Single Ended"	I, Q data only
"Differential"	I, Q and inverse I,Q data (Not available for R&S FSW85)

Remote command:
`INPut:IQ:BALanced[:STATe]` on page 136

High Accuracy Timing Trigger - Baseband - RF
Activates a mode with enhanced timing accuracy between analog baseband, RF and external trigger signals.

Note: Prerequisites for previous models of R&S FSW.
For R&S FSW models with a serial number lower than 103000, special prerequisites and restrictions apply for high accuracy timing:
- To obtain this high timing precision, trigger port 1 and port 2 must be connected via the Cable for High Accuracy Timing (order number 1325.3777.00).
- As trigger port 1 and port 2 are connected via the cable, only trigger port 3 can be used to trigger a measurement.
- Trigger port 2 is configured as output if the high accuracy timing option is active. Make sure not to activate this option if you use trigger port 2 in your measurement setup.
- When you first enable this setting, you are prompted to connect the cable for high accuracy timing to trigger ports 1 and 2. If you cancel this prompt, the setting remains disabled. As soon as you confirm this prompt, the cable must be in place - the firmware does not check the connection. (In remote operation, the setting is activated without a prompt.)

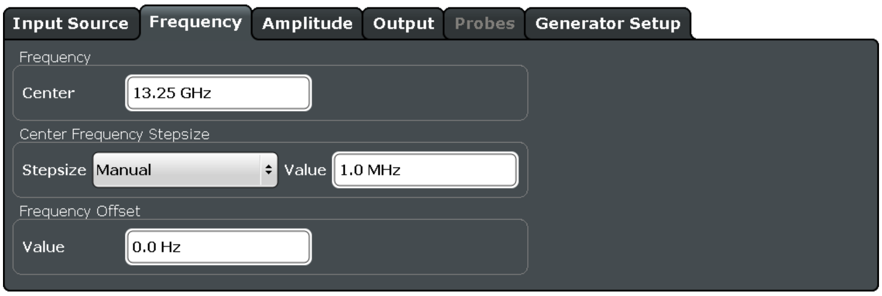
For more information see the R&S FSW I/Q Analyzer and I/Q Input User Manual.

Remote command:
`CALibration:AIQ:HATiming[:STATe]` on page 134

3.4.2 Configuring the Frequency

The "Frequency" tab of the "Input / Output" dialog box contains settings to configure frequency characteristics.

► To access the frequency settings, proceed as follows:
- In the "Configuration Overview", select the "Input / Output" button and then the "Frequency" tab.
- Press the FREQ key and then select the "Frequency Config" softkey.



The frequency characteristics are similar to those available in the Spectrum application. For a comprehensive description of these settings, please refer to the R&S FSW User Manual.

Center Frequency

Defines the frequency of the measured signal.

The possible value range depends on the R&S FSW model you have. See the data sheet for more information about the supported frequency range.

Remote command:
`[SENSe:]FREQuency:CENTer` on page 137

Center Frequency Stepsize

Defines the step size by which the center frequency is increased or decreased when the arrow keys are pressed.

When you use the rotary knob the center frequency changes in steps of only 1/10 of the "Center Frequency Stepsize".

"= Center" Sets the step size to the value of the center frequency and removes the coupling of the step size to span or resolution bandwidth. The used value is indicated in the "Value" field.

"Manual" Defines a fixed step size for the center frequency. Enter the step size in the "Value" field.

Remote command:
`[SENSe:]FREQuency:CENTer:STEP` on page 138

Frequency Offset

Shifts the displayed frequency range along the x-axis by the defined offset.

This parameter has no effect on the instrument's hardware, or on the captured data or on data processing. It is simply a manipulation of the final results in which absolute frequency values are displayed. Thus, the x-axis of a spectrum display is shifted by a constant offset if it shows absolute frequencies, but not if it shows frequencies relative to the signal's center frequency.

A frequency offset can be used to correct the display of a signal that is slightly distorted by the measurement setup, for example.

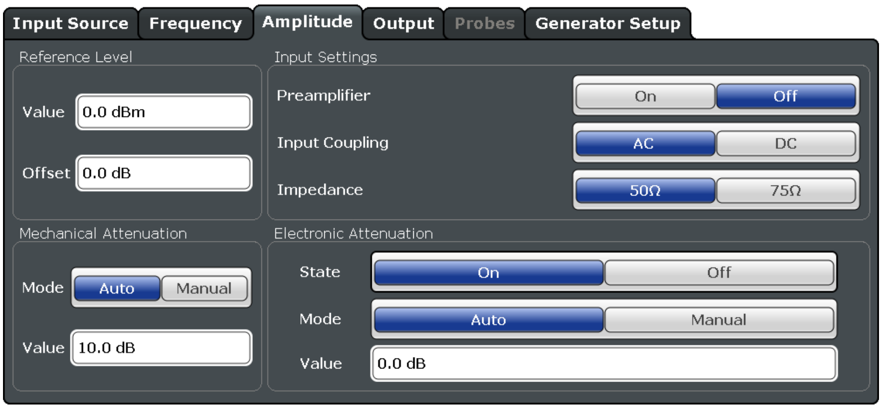
The allowed values range from -100 GHz to 100 GHz. The default setting is 0 Hz.

Remote command:
`[SENSe:]FREQuency:OFFSet` on page 138

3.4.3 Defining Level Characteristics

The "Amplitude" tab of the "Input / Output" dialog box contains settings to configure the signal level characteristics.

► To access the amplitude settings, proceed as follows:
- In the "Configuration Overview", select the "Input / Output" button and then the "Amplitude" tab.
- Press the AMPT key and then select the "Amplitude Config" softkey.



The level characteristics are the same as those available in the Spectrum application. For a comprehensive description of these settings, please refer to the R&S FSW User Manual.

Functions available in the "Amplitude" dialog box described elsewhere:

- "Input Coupling" on page 35
- "Impedance" on page 36

Reference Level

Defines the expected maximum reference level. Signal levels above this value may not be measured correctly, which is indicated by the "IF OVLD" status display.

The reference level is also used to scale power diagrams; the reference level is then used as the maximum on the y-axis.

Since the hardware of the R&S FSW is adapted according to this value, it is recommended that you set the reference level close above the expected maximum signal level to ensure an optimum measurement (no compression, good signal-to-noise ratio).

Remote command:
`DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:RLEVel` on page 139

Shifting the Display (Offset) ← Reference Level

Defines an arithmetic level offset. This offset is added to the measured level. The scaling of the y-axis is changed accordingly.

Define an offset if the signal is attenuated or amplified before it is fed into the R&S FSW so the application shows correct power results. All displayed power level results will be shifted by this value.

The reference level offset takes level offsets into account that occur after the signal has passed through the DUT (usually an amplifier). For level offsets occuring before the DUT, you can define a level offset on the signal generator from within the R&S FSW-K18 user interface.

The setting range is ±200 dB in 0.01 dB steps.

Note, however, that the *internal* reference level (used to adjust the hardware settings to the expected signal optimally) ignores any "Reference Level Offset". Thus, it is important to keep in mind the actual power level the R&S FSW must handle, and not to rely on the displayed reference level (internal reference level = displayed reference level - offset).

Remote command:
`DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:RLEVel:OFFSet` on page 139

Full Scale Level

The full scale level defines the maximum power you can input at the Baseband Input connector without clipping the signal.

- 0.25 V
- 0.5 V
- 1 V
- 2 V

If probes are connected, the possible full scale values are adapted according to the probe's attenuation and maximum allowed power.

Available for parallel capture on the baseband and RF inputs.

Remote command:
`INPut:IQ:FULLscale:LEVel` on page 141

Preamplifier
If the (optional) Preamplifier hardware is installed, a preamplifier can be activated for the RF input signal.

You can use a preamplifier to analyze signals from DUTs with low input power.

For R&S FSW 26 or higher models, the input signal is amplified by 30 dB if the preamplifier is activated.

For R&S FSW 8 or 13 models, the following settings are available:

"Off"	Deactivates the preamplifier.
"15 dB"	The RF input signal is amplified by about 15 dB.
"30 dB"	The RF input signal is amplified by about 30 dB.

Remote command:
`INPut:GAIN:STATe` on page 141
`INPut:GAIN[:VALue]` on page 141

Input Coupling
The RF input of the R&S FSW can be coupled by alternating current (AC) or direct current (DC).

AC coupling blocks any DC voltage from the input signal. This is the default setting to prevent damage to the instrument. Very low frequencies in the input signal may be distorted.

However, some specifications require DC coupling. In this case, you must protect the instrument from damaging DC input voltages manually. For details, refer to the data sheet.

Remote command:
`INPut:COUPling` on page 135

Impedance
For some measurements, the reference impedance for the measured levels of the R&S FSW can be set to 50 Ω or 75 Ω.

75 Ω should be selected if the 50 Ω input impedance is transformed to a higher impedance using a 75 Ω adapter of the RAZ type (= 25 Ω in series to the input impedance of the instrument). The correction value in this case is 1.76 dB = 10 log (75Ω/50Ω).

Remote command:
`INPut:IMPedance` on page 136

Attenuation Mode / Value
The RF attenuation can be set automatically as a function of the selected reference level (Auto mode). This ensures that the optimum RF attenuation is always used. It is the default setting.

By default and when electronic attenuation is not available, mechanical attenuation is applied.

In "Manual" mode, you can set the RF attenuation in 1 dB steps (down to 0 dB). Other entries are rounded to the next integer value. The range is specified in the data sheet. If the defined reference level cannot be set for the defined RF attenuation, the reference level is adjusted accordingly and the warning "Limit reached" is displayed.

NOTICE! Risk of hardware damage due to high power levels. When decreasing the attenuation manually, ensure that the power level does not exceed the maximum level allowed at the RF input, as an overload may lead to hardware damage.

Remote command:
`INPut:ATTenuation` on page 139
`INPut:ATTenuation:AUTO` on page 139

Using Electronic Attenuation
If the (optional) Electronic Attenuation hardware is installed on the R&S FSW, you can also activate an electronic attenuator.

In "Auto" mode, the settings are defined automatically; in "Manual" mode, you can define the mechanical and electronic attenuation separately.

Note: Electronic attenuation is not available for stop frequencies (or center frequencies in zero span) >13.6 GHz.
In "Auto" mode, RF attenuation is provided by the electronic attenuator as much as possible to reduce the amount of mechanical switching required. Mechanical attenuation may provide a better signal-to-noise ratio, however.

When you switch off electronic attenuation, the RF attenuation is automatically set to the same mode (auto/manual) as the electronic attenuation was set to. Thus, the RF attenuation may be set to automatic mode, and the full attenuation is provided by the mechanical attenuator, if possible.

Both the electronic and the mechanical attenuation can be varied in 1 dB steps. Other entries are rounded to the next lower integer value.

For the R&S FSW85, the mechanical attenuation can be varied only in 10 dB steps.

If the defined reference level cannot be set for the given attenuation, the reference level is adjusted accordingly and the warning "Limit reached" is displayed in the status bar.

Remote command:
`INPut:EATT:STATe` on page 140
`INPut:EATT:AUTO` on page 140
`INPut:EATT` on page 140

3.4.4 Using Probes

The "Probes" tab of the "Input / Output" dialog box contains settings to configure and use probes.

Probes are a mandatory part of the test setup if you want to perform measurements measurements that take into account the supply voltage and the current drawn by the PA.

► Select the "Input / Output" button from the "Configuration Overview" dialog box, then select the "Probes" tab.

For more information about the contents of the "Probes" dialog box, please refer to the R&S FSW User Manual.

3.4.5 Configuring Outputs

The "Output" tab of the "Input / Output" dialog box contains settings to configure the various signal outputs available on the R&S FSW.

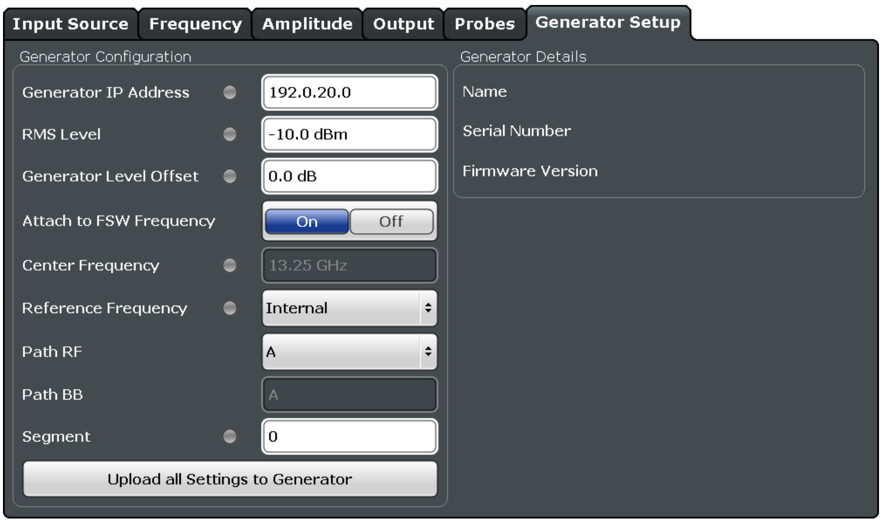
The functionality is the same as in the Spectrum application. For more information about the output functions, please refer to the R&S FSW User Manual.

3.4.6 Controlling a Signal Generator

The "Generator Setup" tab of the "Input / Output" dialog box contains settings to control the signal generator from within the R&S FSW-K18. A remote control connection between the R&S FSW and the signal generator has to be established to be able to do so.

Because a signal generator is (mostly) mandatory in the test setup, these features make measurement configuration as easy as possible. This way, you can control both analyzer and generator from within the application without having to operate the two instruments to configure the measurement.

► To access the generator setup settings, proceed as follows:
In the "Configuration Overview", select the "Input / Output" button and then the "Generator Setup" tab.



State of operation

Most settings have an LED that shows the state of the corresponding setting on the signal generator.

The LED is either grey, green or red:

- Grey LED
  Configuration state unknown (for example when you have not yet started the transmission).
- Green LED

Configuration has been successful. Generator has been configured correctly.

- Red LED
  Configuration has not been successful.
  Check if the connection between analyzer and generator has been established or if the IP address has been stated correctly.

Generator details

The "Generator Details" contain information about the connected signal generator, like the software version or the serial number of the generator.

Updating generator settings

When you change the generator level or frequency in this dialog, the application automatically updates those settings on the generator.

When you use the "Update Generator Setting Manually" button, you can force an update of all generator settings available in this dialog box. Useful when you change the level or frequency on the generator itself. In that case, those settings remain the same in the R&S FSW-K18. To restore the original settings defined within the R&S FSW-K18, use that button and the generator settings will be restored.

Remote command:

`CONFigure:GENerator:SETTings:UPDate` on page 146

Generator IP Address

Defines the IP address of the signal generator connected to the analyzer via LAN.

If you are not sure about the IP address of your generator, kindly ask your IT administrator if he can provide one.

Remote command:

`CONFigure:GENerator:IPConnection:ADDRess` on page 143

`CONFigure:GENerator:IPConnection:LEDState?` on page 144

Generator RMS Level

Defines the RMS level of the signal that should be generated.

When you define the RMS level here, the signal generator is automatically configured to that level.

In addition, you can define a level offset (for example to take external attenuation into account). Note that the level offset is a purely mathematical value and does not change the actual level of the signal at the RF output.

The level offset takes level offsets into account that occur before the signal has passed through the DUT (usually an amplifier). For level offsets occuring after the DUT, define a level offset in the "Amplitude" menu of the signal analyzer.

NOTICE! Risk of damage to the DUT.
RMS levels that are too high may damage or detroy the DUT.

Make sure to keep an eye on the RMS level, especially when defining a level offset, because a level offset changes the displayed value of the RMS level, but not the real RMS level (Displayed RMS Level = Real RMS Level + Level Offset). Thus, the actual RMS level may be higher than the displayed level.

Note: Make sure to always change the generator level from within the R&S FSW-K18 user interface and thus synchronize the level of both instruments.
If you change the generator level on the signal generator, the R&S FSW-K18 won't synchronize the levels and measurement results are going to be invalid.

Remote command:
RMS level: `CONFigure:GENerator:POWer:LEVel` on page 144
`CONFigure:GENerator:POWer:LEVel:LEDState?` on page 144
Level offset: `CONFigure:GENerator:POWer:LEVel:OFFSet` on page 145
`CONFigure:GENerator:POWer:LEVel:OFFSet:LEDState?` on page 145

Attach to R&S FSW Frequency

Turns synchronization of the analyzer and generator frequency on and off.

When you turn this feature on, changing the frequency on the analyzer automatically adjusts the frequency on the generator.

Remote command:
`CONFigure:GENerator:FREQuency:CENTer:SYNC[:STATe]` on page 143

Center Frequency

Defines the frequency of the signal that the generator transmits.

When you turn Attach to R&S FSW Frequency on, any changes you make to the generator frequency are also adjusted on the analyzer.

Remote command:
`CONFigure:GENerator:FREQuency:CENTer` on page 143
`CONFigure:GENerator:FREQuency:CENTer:LEDState?` on page 143

Path RF / BB

Selects the RF signal path of the generator that should be used for signal generation.

Note that the baseband path (which is required for envelope tracking measurements) is always the same as the RF path.

Remote command:
RF path: `CONFigure:GENerator:TARGet:PATH:RF` on page 147
BB path: `CONFigure:GENerator:TARGet:PATH:BB?` on page 146

Selecting a segment in a multi segment waveform file

If you are using a waveform file that contains several different waveforms, you have to select the segment that should be transferred to the signal generator.

Note that the segment that you have selected in the "Generator Setup" has to match the segment selected for the reference signal, regarding the signal characteristics.

Remote command:

`CONFigure:GENerator:SEGMent` on page 145

`CONFigure:GENerator:SEGMent:LEDState?` on page 146

3.5 Triggering Measurements

The R&S FSW-K18 provides functionality to perform triggered measurements.

The "Trigger" dialog box contains settings to configure triggered measurements.

The following trigger sources are supported:

- Free Run
- External
- I/Q Power
- IF Power
- RF Power

► To access the trigger settings, proceed as follows:
  - In the "Configuration Overview", select the "Trigger" button.
  - Press the TRIG key and then select the "Trigger Config" softkey.

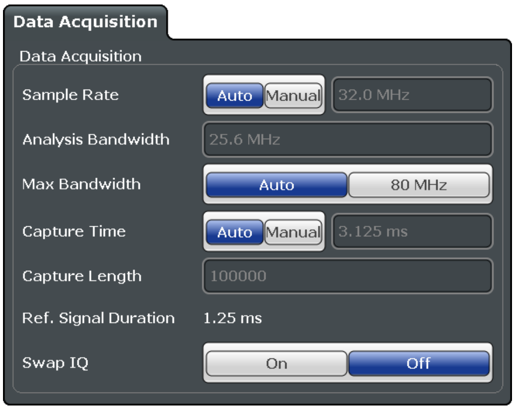
The functionality to configure triggered measurements is similar as that provided in the Spectrum application. For a comprehensive description of the trigger functionality, please refer to the R&S FSW User Manual.

The functionality to configure those trigger sources is similar as that provided in the Spectrum application. For a comprehensive description of the trigger functionality, please refer to the R&S FSW User Manual.

3.6 Configuring the Data Capture

The "Data Acquisition" dialog box contains settings to configure the process of how the application records the signal.

► To access the data acquisition settings, proceed as follows:
  - In the "Configuration Overview", select the "Data Acquisition" button and then the "Data Acquisition" tab.
  - Press the MEAS CONFIG key and then select the "Data Acquisition" softkey.
  - Press the BW key and then select the "Bandwidth Config" softkey.



Configuring the measurement bandwidth

The sample rate defined for data acquisition is the sample rate with which the analyzer samples the amplified signal.

The measurement bandwidth defines the flat, usable bandwidth of the final I/Q data.

The application allows you to adjust both values automatically or manually.

Automatic adjustment

When you select automatic adjustment of sample rate and measurement bandwidth, the application selects a bandwidth that is appropriate for the characteristics of the reference signal and adjusts the sample rate accordingly.

For more information about the reference signal, see chapter 3.3, "Designing a Reference Signal", on page 27.

Manual definition

When you define the sample rate and measurement bandwidth manually, you can select values that you are comfortable with. Because the bandwidth is a function of the sample rate (and vice versa), the application adjusts the values when you change either setting.

The following dependencies apply:

- When you change the sample rate, the application updates the bandwidth accordingly (and vice versa). It also adjusts the capture length to the new values. The capture time remains the same.
- When you change the capture time or capture length, the sample rate and bandwidth remain the same.

Maximum bandwidth

The maximum bandwidth you can use depends on your hardware configuration.

(The following bandwidth extensions are available for the R&S FSW: 160 MHz, 320 MHz, 500 MHz.)

By default, the application automatically determines the maximum bandwidth. When you select a maximum bandwidth other than "Auto", the bandwidth is restricted to that value. When you select the maximum bandwidth manually, make sure that this bandwidth is suited for the signal you are testing. Otherwise, the signal may be distorted and results are no longer valid.

When you are using the baseband input R&S FSW-B71, the maximum bandwidth is always limited to 80 MHz.

If you have no bandwidth extension this setting is not available.

Remote command:
Mode: `TRACe:IQ:SRATe:AUTO` on page 149
Sample Rate: `TRACe:IQ:SRATe` on page 149
Bandwidth: `TRACe:IQ:BWIDth` on page 149
Max. Bandwidth: `TRACe:IQ:WBANd[:STATe]` on page 150
Max. Bandwidth: `TRACe:IQ:WBANd:MBWidth` on page 150

Configuring the measurement time

The measurement time (or capture time) defines the duration of a measurement in which the required number of samples is collected.

The capture length is the number of samples that are captured during the selected measurement time. The capture length is a function of the sample rate and the capture time.

Automatic adjustment

When you select automatic adjustment of capture time, the application selects a capture time that is appropriate for the characteristics of the reference signal.

Length of the reference signal

As orientation, the application shows the length of the reference signal in the corresponding field in the dialog box (➔ "Ref Signal Duration").

For more information about the reference signal, see chapter 3.3, "Designing a Reference Signal", on page 27.

Manual definition

When you define the capture length and time manually, you can select values that you are comfortable with.

However, make sure to define a capture time that is greater than the length of the reference signal - otherwise the application won't be able to analyze the signal correctly.

The following dependencies apply:
- When you change the capture time, the application updates the capture length accordingly (and vice versa). Sample rate and bandwidth remain the same.
- When you change the sample rate or bandwidth, the application updates the capture length accordingly. The capture time remains the same.

Note that the maximum capture time depends on the current measurement bandwidth.

Remote command:
Mode: `[SENSe:]SWEep:TIME:AUTO` on page 148
Time: `[SENSe:]SWEep:TIME` on page 148
Capture length: `[SENSe:]SWEep:LENGth` on page 148
Reference signal: `[SENSe:][REFSig:]TIME?` on page 147

Inverting the I/Q branches

The application allows you to swap the I and Q branches of the signal, if required.

This is useful, for example, when the DUT inverts the real (I) and imaginary (Q) parts of the signal and transfers the signal that way.

Note that the sideband is also inverted when you turn this feature on.

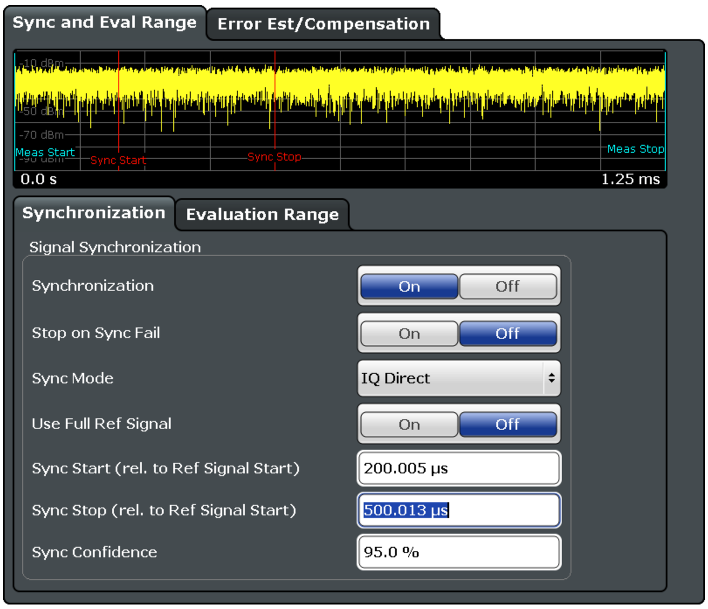
Remote command:
`[SENSe:]SWAPiq` on page 147

3.7 Synchronizing Measurement Data

The R&S FSW-K18 allows you to synchronize the measured signal with the reference signal and provides various features to control synchronization.

Synchronization consists of signal estimation and compensation. After the application has detected the position of the reference signal in the capture buffer, it estimates possible errors in the measured signal (for example the sample error rate or the amplitude droop) by comparing it to the reference signal. The estimated errors can optionally be compensated for.

► To access synchronization settings, proceed as follows:
- In the "Configuration Overview", select the "Sync / Error Est / Comp" button, then the "Sync and Eval Range" tab and then the "Synchronization" tab.
- Press the MEAS CONFIG key, select the "Sync / Error Est / Comp" softkey, then the "Sync and Eval Range" tab and then the "Synchronization" tab.



Turning synchronization of reference and measured signal on and off

During measurements, the application tries to synchronize the measured signal with the reference signal. When no significant correlation between the measured and reference signal can be found, synchronization fails.

However, you can turn synchronization off in case you would like to perform unsynchronized measurements. Note however, that the calculation of some results in the Result Summary requires synchronization. These cannot be calculated when you turn off synchronization.

When you turn off synchronization, the results are always calculated over the complete capture buffer. When synchronization is on, the results are always calculated over the synchronized data range of the capture buffer. Therefore, the result values may be different for unsynchronized measurements, even if you measure the same signal (the result is still valid and correct, though).

Failed synchronization

When you turn on "Stop on Sync Failed", the application automatically aborts the measurement, in case synchronization fails.

Remote command:

`CONFigure:SYNC:STATe` on page 153
`CONFigure:SYNC:SOFail` on page 152

Selecting the synchronization method

The application allows you to select the method with which the application synchronizes the signals with the "Sync Domain" parameter. The following methods are available.

- I/Q Direct
  The I/Q data for the reference signal is directly correlated with the reference and measured signal. The performance of this method will degrade in the presence of a frequency offset between the measured and reference signals.
- I/Q Phase Difference
  Correlation on the phase differentiated I/Q data. This retains phase change information and can handle a frequency offset , but is more sensitive to noise than the "I/Q Direct" method.
- I/Q Magnitude
  Correlation on the magnitude of the I/Q data with no regard for phase information. This method can handle a frequency offset and is less sensitive to noise that the "I/Q Phase Diff" method, but is only useful with amplitude modulated signals.
- Trigger
  It is assumed that the capture is triggered at the start of the reference waveform. Only minimal correlation is performed to account for trigger jitter. This is the fastest synchronization method.

Remote command:

`CONFigure:SYNC:DOMain` on page 152

Defining a synchronization confidence level

The synchronization confidence level ("Sync Confidence") is a percentage that describes how similar (or correlated) reference and measured signal need to be in order for synchronization to be successful.

A value of 0 % means that synchronization will always be successful even if the signals are not correlated at all. However, results that rely on a good synchronization (like the EVM) do contain reasonable values in that case. A value of 100 % means that the signals are identical (in that they are linearily dependent).

The cross-correlation is calculated over all samples in the capture buffer (or the estimation range, if you have defined one).

As soon as the cross-correlation coefficient falls below the confidence level you have defined, synchronization is no longer successful.

Remote command:
`CONFigure:SYNC:CONFidence` on page 152

Defining the estimation range

The estimation range has several effects on the synchronization process.

- It defines which part of the reference signal is used for cross-correlation within the capture buffer in order to align the reference and measured signals.
- It defines which part of the reference signal is used for error estimation.

By default, the application estimates over the complete reference signal. However, you can also estimate over a given range in the capture buffer only. In that case, turn off the "Use Full Ref Signal" feature. When this is off, the "Eval Start" and "Eval Stop" fields become available. The allowed values are offsets relative to the beginning of the capture buffer (0 s). The highest offset possible depends on the size of the capture buffer.

Defining an estimation range is useful in the following cases.

- If you want to limit the estimation to a specific part of the signal, for example if the signal contains a preamble or midamble.
- If you want to limit the estimation to the ON part of a TDD signal.
- If you want to increase the measurement speed in case of relatively long signals, for example an LTE signal.

On the downside, limiting the estimation range leads to a higher empirical variance of the results.

In the preview pane displayed in the dialog box, the currently defined estimation range is represented by two red vertical lines.

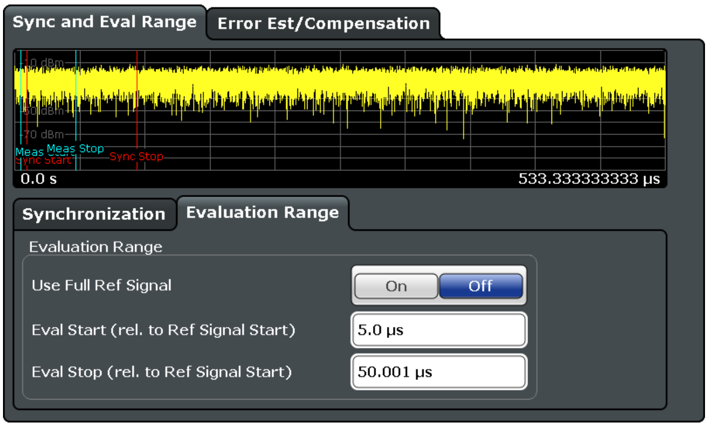
Tip: You can also move the corresponding lines in the preview pane with your fingers to a new position. However, this is not as accurate as entering a number into the input field.

Remote command:
`CONFigure:ESTimation:FULL` on page 151
`CONFigure:ESTimation:STARt` on page 151
`CONFigure:ESTimation:STOP` on page 151

3.8 Evaluating Measurement Data

The R&S FSW-K18 allows you to define the time frame in the reference signal used to evaluate and calculate the measurement results.

► To access the evaluation range settings, proceed as follows:
- In the "Configuration Overview", select the "Sync / Error Est / Comp" button, then the "Sync and Eval Range" tab and then the "Evaluation" tab.
- Press the MEAS CONFIG key, select the "Sync / Error Est / Comp" softkey, then the "Sync and Eval Range" tab and then the "Evaluation" tab.



Defining the evaluation range
The evaluation range defines the data range in the capture buffer over which the application calculates the measurement results.

By default, the application calculates the results over the complete capture buffer. If synchronization has been succesful, the application calculates the results over the capture buffer range in which the reference signal has been found. If you have turned off synchronization or if it hasn't been successful, the complete capture buffer is used to calculate the remaining results.

Example:

The capture buffer is 30 ms long, the reference signal starts at 9 ms and is 10 ms long. In case of successful synchronization, the evaluation range starts at 9 ms and ends at 19 ms. If synchronization has been turned off, the evaluation range is the full capture buffer.

However, you can also select a particular data range within the reference signal. In that case, turn off the "Use Full Ref Signal" feature. When this is off, the "Eval Start" and "Eval Stop" fields become available. The allowed values are offsets relative to the beginning of the reference signal (0 s). The highest offset possible depends on the length of the reference signal.

Example:

The situation is as described above (30 ms capture buffer, 10 ms reference signal). Let's say you want to evaluate milliseconds 2 to 6 of the reference signal. In that case, you would have to define a start offset of 11 ms (the reference signal starts at 9 ms, plus the first 2 ms you are not interested in = 11 ms) and a stop offset of 15 ms (9 ms + 6 ms).

In the preview pane displayed in the dialog box, the currently defined evaluation range is represented by two blue vertical lines.

Tip: You can also move the corresponding lines in the preview pane with your fingers to a new position. However, this is not as accurate as entering a number into the input field.

Remote command:

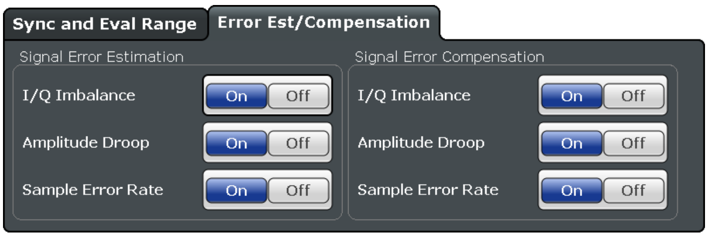
CONFigure:EVALuation:FULL on page 153
CONFigure:EVALuation:STARt on page 154
CONFigure:EVALuation:STOP on page 154

3.9 Estimating and Compensating Signal Errors

The application allows you to estimate possible undesired effects in the signal, and, if there are any, also compensate these effects.

► To access error estimation and compensation, proceed as follows:

- In the "Configuration Overview", select the "Sync / Error Est / Comp" button and then the "Error Est / Compensation" tab.
- Press the MEAS CONFIG key, select the "Sync / Error Est / Comp" softkey and then the "Error Est / Compensation" tab.



Configuring error estimation and compensation

When you turn on error estimation only, the results are not compensated for the corresponding errors.

When you turn on error compensation, the displayed results are also corrected by the estimated errors. Note that in that case, the signal might look better than it actually is.

Compensation without estimation is not possible.

You can estimate and compensate the following effects

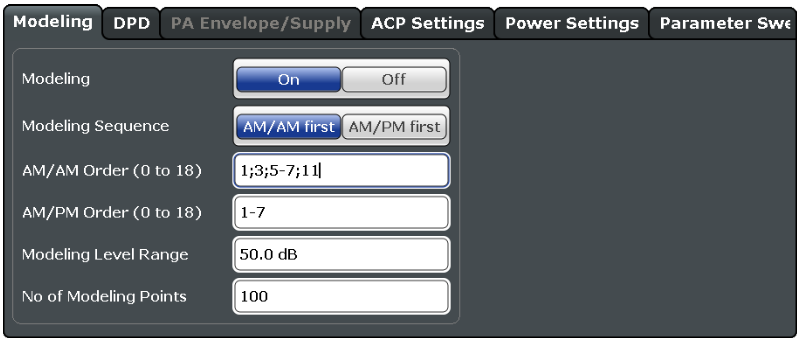
- **I/Q Imbalance**: combined effect of amplitude and phase error.

- **Amplitude Droop**: decrease of the signal power over time in the transmitter.
- **Sample Error Rate**: difference between the sample rate of the reference signal and the measured signal.

3.10 Applying System Models

The R&S FSW-K18 allows you to calculate a polynomial model that describes the characteristics of the DUT based on the input signal and the output signal of the amplifier.

► To access system modeling settings, proceed as follows:
  - In the "Configuration Overview", select the "Measurement" button and then the "Modeling" tab.
  - Press the MEAS CONFIG key and then select the "Meas Config" softkey and then the "Modeling" tab.



Turning system modeling on and off
The R&S FSW-K18 provides functionality to calculate a mathematical model that describes the properties of the DUT.

Using a model is useful to observe and estimate the behavior of the amplifier and, if necessary, adjust the DUT behavior. The application supports memory-free polynomial models to the 18th degree.

The following diagrams contain traces that show the model. These traces are calculated by using the model function on the reference signal.
- AM/AM
- AM/PM

Note that the model traces are also the basis for the DPD functionality available in the R&S FSW-K18.

When the characteristics of the modeled signal matches those of the measured signal, the model describes the DUT behavior well. If not, you can try to get a better result by adjusting the model properties.

When you turn on modeling, the application shows an additional trace in the graphical result displays. This trace corresponds to the signal characteristics after the model has been applied to the reference signal.

Selecting the modeling sequence

The modeling sequence selects the sequence in which the models are calculated. The application then either calculates the AM/AM model before calculating the AM/PM model (default), or vice versa.

Remote command:
`CONFigure:MODeling[:STATe]` on page 159
`CONFigure:MODeling:SEQuence` on page 158

Selecting the degree of the polynomial

In addition to the type of curve you can also select the order of the polynomial model.

The order of the model define the degree, complexity and number of terms in the polynomial model. In general, a polynomial of the N^{th} degree looks like this.

$y = a_0 + a_1x + a_2x^2 + \ldots + a_Nx^N$

The degree of the model is defined by N (as an index or exponent). The higher the order, the more complex the calculation and the longer it takes to calculate the model. Higher models do not necessarily lead to better fitting model curves.

Note that the nonlinear effects consume an additional bandwidth proportional to 2 times the number of odd factors in the polynom, excluding the linear one.

Example:

If the signal bandwidth is 1 MHz and the highest degree is 5, the bandwidth of the resulting signal is increased by 2 times 2 (because there are the variables a_3 and a_5) times 1 MHz which are 4 MHz and leading to a total signal bandwidth of 5 MHz (1 MHz + 4 MHz). The configured recording bandwidth must be at least 5 MHz to record all nonlinear effects generated by the DUT.

Tip: To select several polynomial degrees that should be applied, you can either:

- Define a range of degrees (e.g. "0 - 5", in that case the application applies all degrees in that range).
- Define a set of individual degrees only (e.g. "1;3;5;7", in that case the application applies those degrees only). Note that the "." key on the front panel draws the ";" character.
- Define a combination of the methods mentioned above (e.g. "1;3;5-7")

Remote command:
AM/AM: `CONFigure:MODeling:AMAM:ORDer` on page 157
AM/PM: `CONFigure:MODeling:AMPM:ORDer` on page 158

Defining the modeling range

By default, the R&S FSW-K18 applies the model to the complete signal. Most of the time, however, it is sufficient to apply the model to a small number of samples covering the linear level range and the peak level of the amplifier.

With the "Modeling Level Range" setting, you can use a smaller level range to apply the model to and still yield valid results. When you limit the level range, only samples with levels between peak level and "peak level minus modeling level range value" are taken into account during the calculation of the model. Note that the modeling range is also the range the DPD is applied to.

In addition, you can define the number of points on the curve that the application uses to calculate the model. The selected points are spaced equidistant. Using less modeling points further speeds up measurement times (but may reduce the quality of the model if set too low).

Remote command:
Range: `CONFigure:MODeling:LRANge` on page 158
Points: `CONFigure:MODeling:NPOints` on page 158

3.11 Applying Digital Predistortion

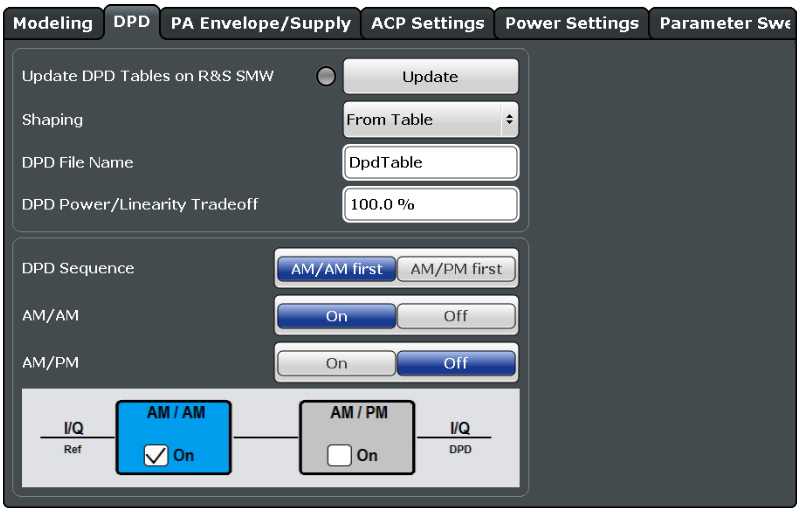
Digital predistortion (DPD) is one of the methods used to improve the efficiency of RF power amplifiers. The R&S FSW-K18 features functionality to deliberately take digital predistortion into account.

There are several known models used to describe distortions. This implementation focuses on the following two types of distortion:

- the AM/AM (amplitude-to-amplitude) distortion and
- the AM/PM (amplitude-to-phase) distortion.

Requirements
Using the predisortion functionality requires an R&S SMW equipped with option R&S SMW-K541.

► To access the DPD settings, proceed as follows:
  - In the "Configuration Overview", select the "Measurement" button and then the "DPD" tab.
  - Press the MEAS CONFIG key, select the "Meas Settings" softkey and then the "DPD" tab.



Selecting the DPD shaping method

The application provides several ways for DPD calculation (or shaping).

- "From Table"
  Shapes the DPD function based on a table that contains the correction values required to predistort the signal.
  The calculation of the table is based on the AM/AM and AM/PM polynomial models.
  For more information about the contents and usage of the shaping table, please refer to the documentation of the R&S SMW.
  You can define a file name for the DPD table in the corresponding field.
- "From Polynomial"
  Shapes the DPD function based on the polynomial defined for the system model. Compared to DPD based on a shaping table, this method does not transfer a list with correction values. Instead, the application transfers the polynomial coefficients of the polynomial function used for the correction.
  For more information see chapter 3.10, "Applying System Models", on page 55.

You can update the DPD shaping on the R&S SMW comfortably with the "Update" button.

Remote command:
Mode: `CONFigure:DPD:SHAPing:MODE` on page 161
Table name: `CONFigure:DPD:FNAMe` on page 160

Selecting the order of model calculation

The application allows you to calculate either the AM/AM distortion, the AM/PM distortion or both simultaneously. You can turn correction of the distortion models on and off in the corresponding fields.

In case you want to predistort both the AM/AM distortion and the AM/PM distortion simultaneously, you can select the order in which the curves are calculated and applied to the I/Q signal on the R&S SMW.

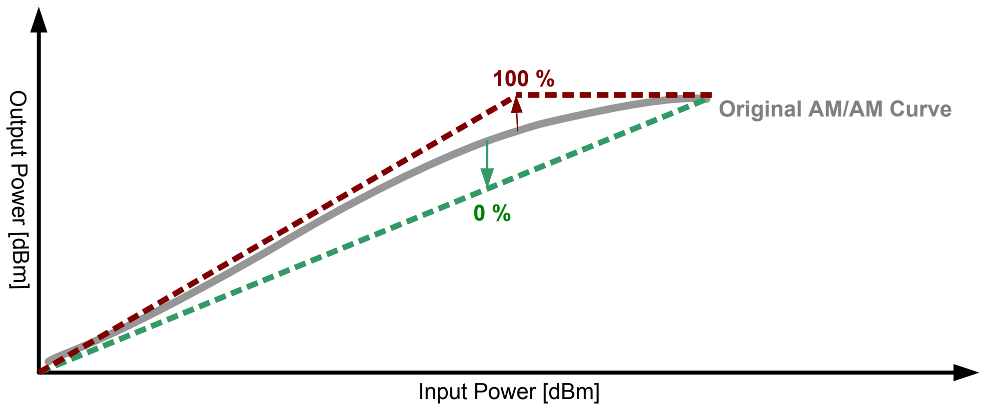
- AM/AM First
  Calculates the AM/AM first, then calculates the AM/PM based on the signal that has already been corrected by its AM/AM distortions.
- AM/PM First
  Calculates the AM/PM first, then calculates the AM/AM based on the signal that has already been corrected by its AM/PM distortions.

Note: the DPD sequence is displayed by the diagram that is part of the dialog box.

Remote command:
AM/AM state: `CONFigure:DPD:AMAM[:STATe]` on page 159
AM/PM state: `CONFigure:DPD:AMPM[:STATe]` on page 160
Both: `CONFigure:DPD:AMXM[:STATe]` on page 160
Calculation order: `CONFigure:DPD:SEQuence` on page 160

DPD Power / Linearity Tradeoff
The "DPD Power / Linearity Tradeoff" describes the effects of the DPD on the amplifier characteristics.

When you define a tradeoff of 0 %, the DPD aims for the best linearity (green line in the illustration below). When you increase the tradeoff value, the DPD aims for an optimization of the output power at the expense of linearity. In the ideal case (red line), the DPD affects the amplifier characteristics in a way that the best output power is achieved.



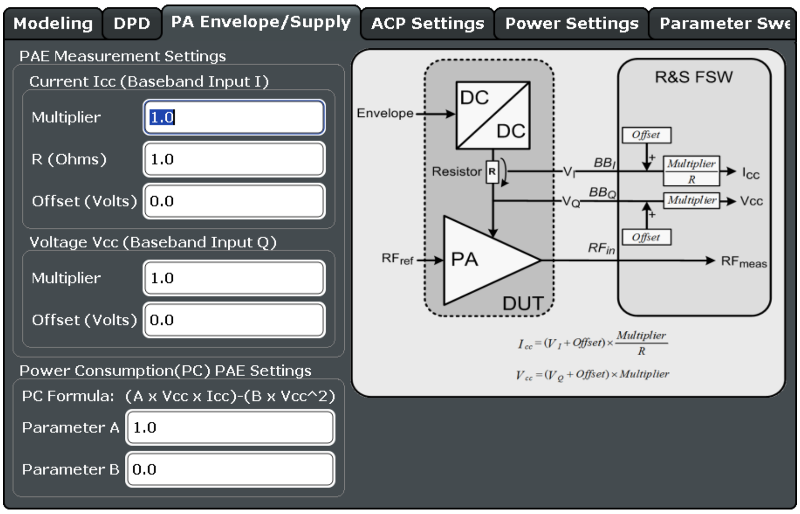
Remote command:
`CONFigure:DPD:TRADeoff` on page 161

3.12 Configuring Envelope Measurements

When you perform measurements on power amplifiers supporting envelope tracking, you have to describe several characteristics of the measurement equipment in order to get valid results.

► To access the envelope settings, proceed as follows:

- In the "Configuration Overview", select the "Measurement" button and then the "PA Envelope / Supply" tab.
- Press the MEAS CONFIG key, select the "Meas Settings" softkey and the "PA Envelope / Supply" tab.



Configuring PAE measurements (Power Added Efficiency)

When you are testing amplifiers that support envelope tracking, the Power Added Efficiency (PAE) of the system is the value that characterizes its performance.

To calculate the PAE, you have to measure the supply voltage and current drawn by the power amplifier. The PAE is calculated according to the following equation:

PAE = (RF Output Power - RF Input Power) / DC Power

with DC Power = Voltage * Current

Measuring the voltage and current requires additional equipment and components in the test setup. For valid measurement results, you have to define the characteristics for those components.

Required components

One way to measure the voltage is to use a probe. The voltage is measured on the Q channel of the baseband input provided by the optional baseband hardware option.

One way to measure the current is to use a shunt resistor and another probe. The current is measured on the I channel of the baseband input provided by the optional baseband hardware option.

For both types of components, you have to accurately define their characteristics and behavior.

Measuring current

When using a shunt resistor to measure the current, you have to define the **resistance R** of the shunt resistor you are using. The resistance is a value with the unit Ω.

The test setup may also have additional characteristics that have to be taken into account (for example those of passive probes). You can take those into account via the **multiplier**. The multiplier is a value without unit. (When you are using an active probe from Rohde & Schwarz, you do not have to change the multiplier, because it is automatically detected by the Amplifier application.)

In addition, you have to compensate the DC offset of active probes. The DC offset is described by the **offset** value, which differs depending on the probe you are using. The offset value has to be measured.

Measuring voltage

To measure the voltage, you also have to define the **multiplier** (to take the attenuation of passive probes into account) as well as the **offset** (to compensate the DC offset of active probes).

Note that entering wrong values for these parameters yields invalid measurement results. Generally speaking, the multiplier multiplies the results by a certain value, the offset is added to the results.

These settings are available when you turn on the baseband input.

Remote command:
See chapter 5.6.12, "Configuring Envelope Tracking", on page 162

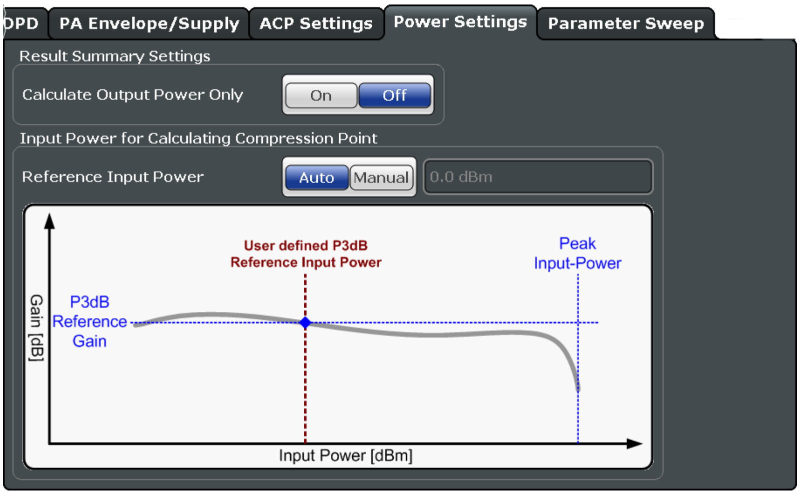
Parameter A / B
Undocumented feature.

Remote command:
`CONFigure:PAE:PCONsumption[:PARameter]:A`
`CONFigure:PAE:PCONsumption[:PARameter]:B`

3.13 Configuring Power Measurements

The R&S FSW-K18 features functionlity to configure measurements that determine power characteristics of an amplifier.

► To access power measurement settings, proceed as follows:
- In the "Configuration Overview", select the "Measurement" button and then the "Power Settings" tab.
- Press the MEAS CONFIG key and then select the "Meas Config" softkey and then the "Power Settings" tab.



Evaluating only the DUT output power

In case you are only interested in the output power of the amplifier, the application allows you to turn off the calculation of all results except the output power.

Doing this has the advantage of speeding up the measurement considerably.

When you turn on the "Calculate Output Power Only" feature, the application only calculates the output power of the amplifier. All other results (numerical or graphical) are not available in that case.

Remote command:
`CONFigure:POWer:RESult:PONLy[:STATe]` on page 166

Configuring compression point calculation

The application evaluates three compression points. The compression points represent the input power where the gain of the amplifier deviates by a certain amount from a reference point on the gain curve (1 dB, 2 dB and 3 dB).

Because these compression points are relative values, you have to define the reference gain.

There are two ways to get the reference gain: automatically or manually.

In case of **manual** specification of the reference gain, the reference point is the gain at a certain input power (which you can define in the "Reference Input Power" input field).

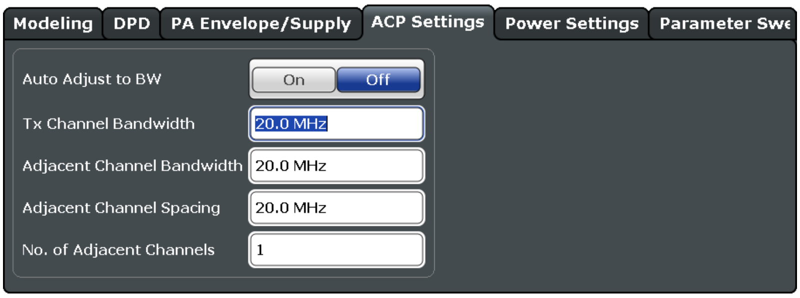
In case of **automatic** calculation of the reference gain, the reference gain is the average gain that has been measured (the average gain is a result shown in the Numeric Result Summary).

Remote command:
Method: `CONFigure:POWer:RESult:P3DB[:STATe]` on page 166
Input power: `CONFigure:POWer:RESult:P3DB:REFerence` on page 166

3.14 Configuring Adjacent Channel Power (ACP) Measurements

The R&S FSW-K18 allows you to define the basic characteristics of the Tx channel and neighboring channels when you perform ACP measurements.

► To access ACP settings, proceed as follows:
- In the "Configuration Overview", select the "Measurement" button and then the "ACP Settings" tab.
- Press the MEAS CONFIG key and then select the "Meas Config" softkey and then the "ACP Settings" tab.



Selecting the measurement bandwidth
When you perform an ACP measurement, it is important to select a measurement bandwidth that is large enough to capture all channels that should be evaluated in the ACP measurement.

The application allows you to automatically adjust the measurement bandwidth to the bandwidth occupied by all channels evaluated in the ACP measurement. To do so, turn on the "Auto Adjust Acquisition Bandwidth" function.

Note that you also have to turn on automatic bandwidth selection in the "Data Acquisition" dialog box in order to adjust the measurement bandwidth to the ACP configuration.

If you define the bandwidth manually, make sure to take one that is large enough to capture all channels. Otherwise, measurement results will not be evaluated. Make also sure that the R&S FSW you are using can actually handle the bandwidth occupied by the transmission and adjacent channels. For larger bandwidths one of the I/Q bandwidth extensions might be necessary (refer to the datasheet for a complete list of available bandwidth extensions).

Remote command:
`[SENSe:]POWer:ACHannel:AABW` on page 164

Defining characteristics of the transmission channel

The ACP measurement requires you to define various basic characteristics of the channels evaluated in the measurement.

For the transmission (Tx) channel, you can define its bandwidth in the "Tx Channel Bandwidth" input field.

Remote command:
[SENSe:]POWer:ACHannel:BANDwidth[:CHANnel] on page 165

Defining characteristics of the adjacent channels

The ACP measurement requires you to define various basic characteristics of the channels evaluated in the measurement.

The application supports measurements on up to three pairs of neighboring channels. Changing the "Number of Adjacent Channels" adds or removes neighboring channels to the left and right side of the Tx channel.

Note that the first adjacent channel on either side of the transmission channel is labeled "Adjacent Channel" in the ACP result display, while the subsequent neighboring channels are labeled "Alternate Channel <x>".

All neighboring channels have the same basic configuration: bandwidth and channel spacing. The "Adjacent Channel Bandwidth" defines the bandwidth of the neighboring channels. The "Channel Spacing" is the distance between the center frequency of one adjacent channel to the center frequency of the next adjacent channel.

Remote command:
Number of channels: [SENSe:]POWer:ACHannel:ACPairs? on page 164
Bandwidth: [SENSe:]POWer:ACHannel:BANDwidth:ACHannel on page 165
Channel spacing: [SENSe:]POWer:ACHannel:SPACing:ACHannel on page 165

3.15 Configuring the Parameter Sweep

The Parameter Sweep is a measurement that allows you to compare a result (that you can select arbitrarily) against two other parameters. The advantage of the Parameter Sweep is that it controls the signal generator and the analyzer, and automatically changes the signal characteristics (for example the frequency) without you having to do those changes manually. In addition, it combines the results in a single and well arranged diagram and / or numerical result display (→ Parameter Sweep).

Example:

In the default state, the application compares the EVM against the frequency and the generator power.

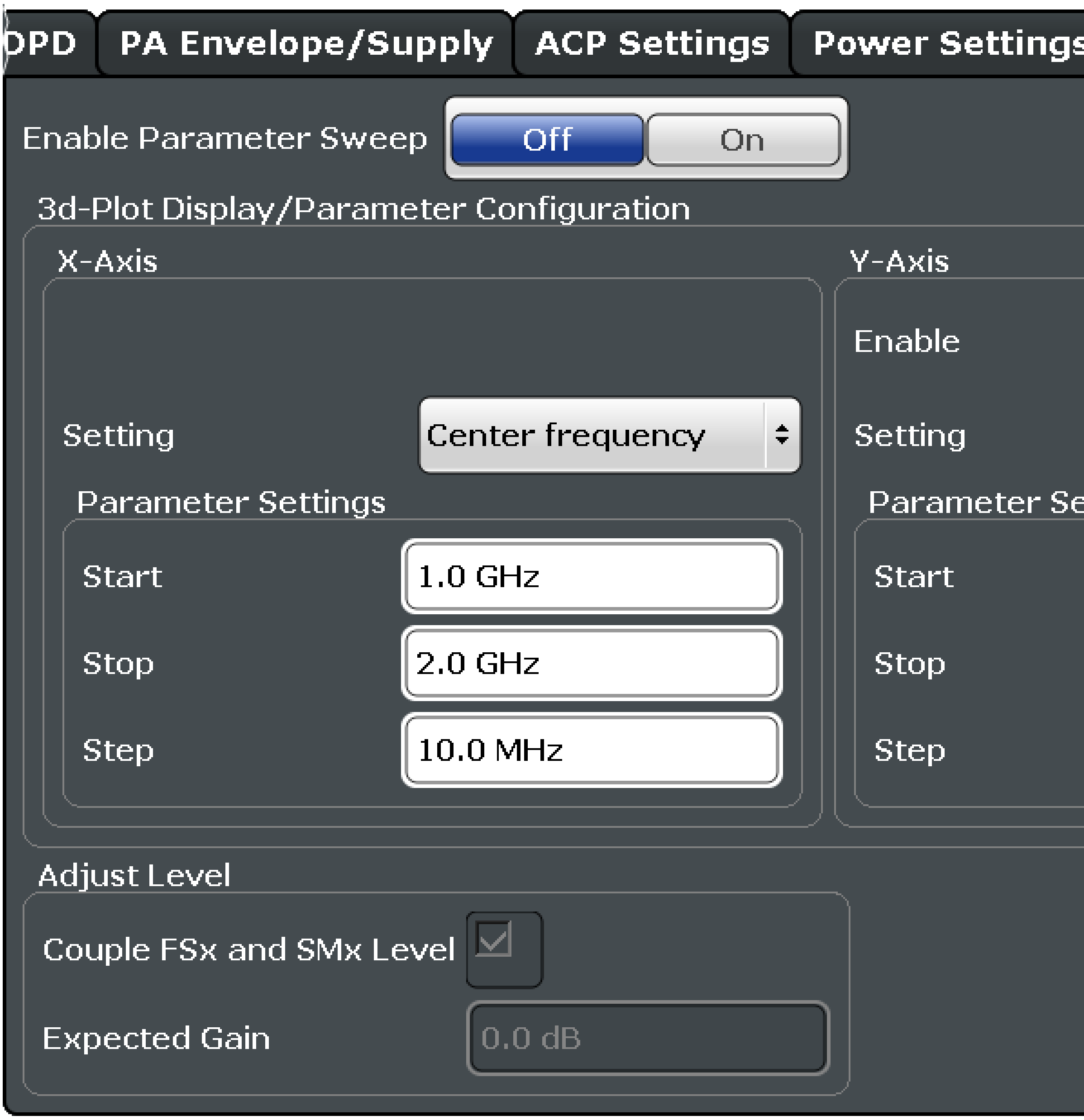
In that case, the application first performs a measurement on the first frequency for each generator output level in the defined range. When this is done, the measurement continues to measure all power levels on the second frequency and so on.

Frequency range: 10 MHz to 20 MHz, stepsize 1 MHz. Output level range: -10 dBm to 0 dBm, stepsize: 1 dB.

- 1st measurement: 10 MHz with a generator output level of -10 dBm.
- (...)
- 11th measurement: 10 MHz with a generator output level of 0 dBm.
- 12th measurement: 11 MHz with a generator output level of -10 dBm.
- (...)
- 22nd measurement: 11 MHz with a generator output level of 0 dBm.
- (...)
- nth measurement: 20 MHz with a generator output level of 0 dBm.

The configuration you have made affects the number of measurement that will be performed. This also has an effect on the overall measurement time of the parameter sweep.

► To access the parameter sweep settings, proceed as follows:
   - In the "Configuration Overview", select the "Measurement" button and then the "Parameter Sweep" tab.
   - Press the MEAS CONFIG key and then select the "Meas Config" softkey and then the "Parameter Sweep" tab.



Turning the parameter sweep on and off

Before you can use the Parameter Sweep functionality, you have to turn it on separately.

When you turn it on, the application starts the Parameter Sweep in single sweep mode (RUN SGL and RUN CONT both start the Parameter Sweep in that case). When the Parameter Sweep is on, other measurements are not possible, and vice versa.

Turning on the Parameter Sweep also expands the channel bar by several labels that carry information about the progress of the Parameter Sweep.

Remote command:
`CONFigure:PSWeep[:STATe]` on page 168

Selecting the data to be evaluated during the Parameter Sweep
When you are performing a Parameter Sweep, you can compare an arbitrary result against one or two arbitrary parameters.

Depending on your selection, the R&S FSW-K18 changes the values of the selected parameters on the signal generator during the measurement, and calculates the result for each combination of values.

Note that when you open more than one instance of the Parameter Sweep, the application applies the selected parameters to all instances (the displayed results on the other hand, can be different for each instance).

- Center Frequency
  Controls the frequency of the signal generator.
- Generator Power
  Controls the output power of the signal generator.
- Envelope to RF Delay
  Controls the delay between the envelope and the RF signal on the signal generator.
- Envelope Bias
  Controls the envelope bias on the signal generator.

You can define the scope of the measurement by adjusting the start and stop values for both parameters, and assign a certain stepsize. Based on these values, the R&S FSW-K18 changes the generator setup after each individual measurement.

The second parameter is not mandatory. You can turn it off with the "Y-Axis Enable" function. In that case, the Parameter Sweep is represented in a two-dimensional diagram (for example the EVM against the frequency).

Example:

When you define a level range from 0 dBm (start value) to 10 dBm (stop value) with a stepsize of 1 dB, the Parameter Sweep would perform 11 measurement on a single frequency.

When you additionally define a frequency range between 10 MHz and 20 MHz, and a stepsize of 1 MHz, the total number of measurements would be 121 (11 power level measurements on each of the 11 frequencies).

Remote command:
chapter 5.6.15, "Configuring Parameter Sweeps", on page 167

Synchronizing the levels of signal generator and analyzer
When you sweep the output level of the generator, make sure to synchronize the reference level of the analyzer and the RMS level of the generator to avoid damage to the RF input of the analyzer (→ "Couple FSx and SMx Level"). When you do so, the application automatically adjusts the reference level of the analyzer to the output level of the generator.

Note that it is mandatory to define the "Expected Gain" of the DUT. Otherwise the synchronization between the levels might fail or lead to invalid results.

NOTICE! Risk of damage to the RF input of the analyzer.
Make sure to define the correct "Expected Gain". Otherwise the gain of the amplifier will not be taken into account during the level changes on signal analyzer and generator, which in turn might lead to a high level signal damaging or destroying the RF input mixer of the analyzer.

With a correct "Expected Gain" value, however, the application is able to attenuate the signal accordingly.

Remote command:
Synchronization state: `CONFigure:PSWeep:ADJust:LEVel[:STATe]` on page 167
Expected gain: `CONFigure:PSWeep:EXPected:GAIN` on page 167

4 Analysis

The R&S FSW-K18 provides several tools to get more information about the results.

Most of these tools work similar to those available in the Spectrum application. For more information about these tools, please refer to the R&S FSW User Manual.

4.1 Configuring Traces

The R&S FSW-K18 provides several tools to configure and evaluate traces.

4.1.1 Selecting the Trace Information

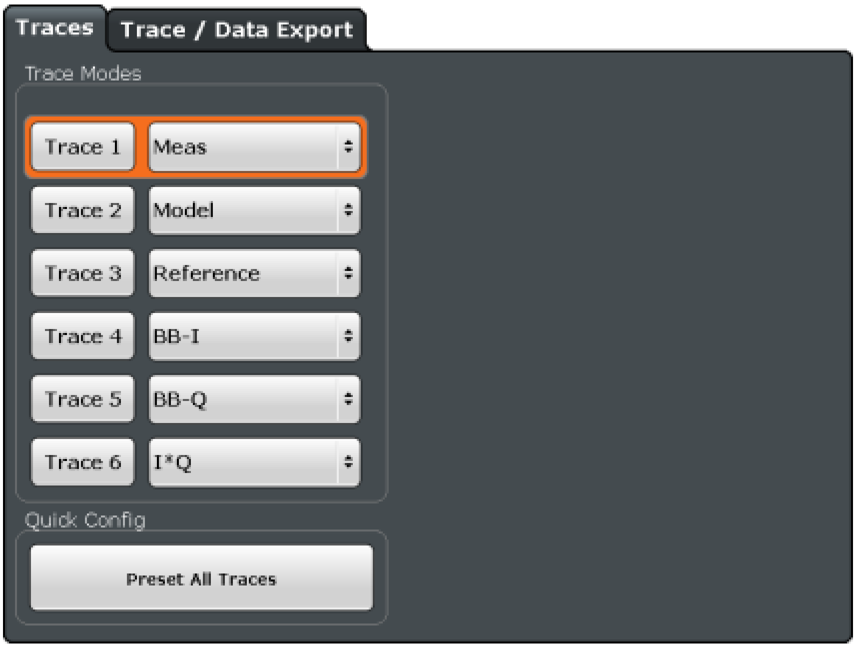
Each result display contains one or several traces specific to the corresponding result type.

The number of traces available for each result display and the information these traces provide are described in chapter 2, "Performing Amplifier Measurements", on page 10.

Remote command:

`DISPlay[:WINDow<n>]:TRACe<t>:MODE` on page 171

► To access the trace configuration, proceed as follows:
Press the TRACE key and select the "Trace Config" softkey.



Restoring default traces

You can press the "Preset All Traces" button anytime to restore the default trace configuration for each result display.

4.1.2 Exporting Traces

The functionality to export traces is the same as in the Spectrum application.

For more information, please refer to the R&S FSW User Manual.

4.2 Using Markers

The R&S FSW-K18 provides four markers in most result displays.

4.2.1 Configuring Individual Markers

The functionality to position markers and query their position is similar to the marker functionality available in the Spectrum application.

► To access markers, proceed as follows:
- In the "Configuration Overview", select the "Result Configuration" button and then the "Markers" or "Marker Settings" tab.
- Press the MKR key and select the "Marker Config" softkey.

Availability of markers

The "Markers" and "Marker Settings" tabs are available for result displays that support markers.

If the tabs are unavailable, make sure to select a result display that actually supports markers from the "Specifics for:" dropdown menu (for example the Spectrum FFT result display).

Note also that the R&S FSW-K18 does not support more than four markers in any result display.

Selected Marker

Marker name. The marker which is currently selected for editing is highlighted orange.

Remote command:
Marker selected via suffix <m> in remote commands.

Marker State

Activates or deactivates the marker in the diagram.

Remote command:
CALCulate<n>:MARKer<m>[:STATe] on page 176
CALCulate<n>:DELTamarker<m>[:STATe] on page 175

Marker Position (X-value)

Defines the position (x-value) of the marker in the diagram.

Remote command:
CALCulate<n>:MARKer<m>:X on page 177
CALCulate<n>:DELTamarker<m>:X on page 175

Marker Type

Toggles the marker type.

The type for marker 1 is always "Normal", the type for delta marker 1 is always "Delta". These types cannot be changed.

Note: If normal marker 1 is the active marker, switching the "Mkr Type" activates an additional delta marker 1. For any other marker, switching the marker type does not activate an additional marker, it only switches the type of the selected marker.

"Normal" A normal marker indicates the absolute value at the defined position in the diagram.

"Delta" A delta marker defines the value of the marker relative to the specified reference marker (marker 1 by default).

Remote command:
`CALCulate<n>:MARKer<m>[:STATe]` on page 176
`CALCulate<n>:DELTamarker<m>[:STATe]` on page 175

Reference Marker
Defines a marker as the reference marker which is used to determine relative analysis results (delta marker values).

Remote command:
`CALCulate<n>:DELTamarker<m>:MREF` on page 174

Assigning the Marker to a Trace
The "Trace" setting assigns the selected marker to an active trace. The trace determines which value the marker shows at the marker position. If the marker was previously assigned to a different trace, the marker remains on the previous frequency or time, but indicates the value of the new trace.

If a trace is turned off, the assigned markers and marker functions are also deactivated.

Remote command:
`CALCulate<n>:MARKer<m>:TRACe` on page 176

All Markers Off
Deactivates all markers in one step.

Remote command:
`CALCulate<n>:MARKer<m>:AOFF` on page 176

Marker Table Display
Defines how the marker information is displayed.

"On" Displays the marker information in a table in a separate area beneath the diagram.

"Off" Displays the marker information within the diagram area.

"Auto" (Default) Up to two markers are displayed in the diagram area. If more markers are active, the marker table is displayed automatically.

Remote command:
`DISPlay:MTABle` on page 173

4.2.2 Positioning Markers

Peak Search
Sets the selected marker/delta marker to the maximum of the trace. If no marker is active, marker 1 is activated.

Remote command:
`CALCulate<n>:MARKer<m>:MAXimum[:PEAK]` on page 180
`CALCulate<n>:DELTamarker<m>:MAXimum[:PEAK]` on page 178

Search Next Peak
Sets the selected marker/delta marker to the next (lower) maximum of the assigned trace. If no marker is active, marker 1 is activated.

Remote command:
`CALCulate<n>:MARKer<m>:MAXimum:NEXT` on page 179
`CALCulate<n>:DELTamarker<m>:MAXimum:NEXT` on page 178

Search Minimum
Sets the selected marker/delta marker to the minimum of the trace. If no marker is active, marker 1 is activated.

Remote command:
`CALCulate<n>:MARKer<m>:MINimum[:PEAK]` on page 180
`CALCulate<n>:DELTamarker<m>:MINimum[:PEAK]` on page 179

Search Next Minimum
Sets the selected marker/delta marker to the next (higher) minimum of the selected trace. If no marker is active, marker 1 is activated.

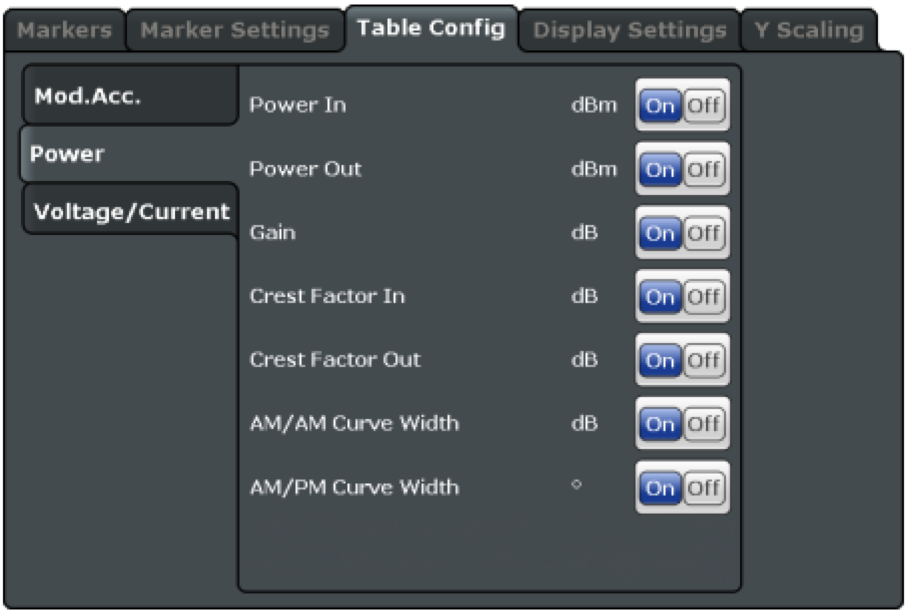
Remote command:
`CALCulate<n>:MARKer<m>:MINimum:NEXT` on page 180
`CALCulate<n>:DELTamarker<m>:MINimum:NEXT` on page 179

4.3 Customizing Numerical Result Tables

The R&S FSW-K18 allows you to customize the contents of the Result Summary and the Parameter Sweep Table.

By default, the application shows all supported numerical results in the result tables (Result Summary and Parameter Sweep Table). However, you can add or remove results as you like.

► To access the table configuration, proceed as follows:
  - In the "Configuration Overview", select the "Result Configuration" button and then the "Table Config" tab.



Accessing the "Table Config" tab

Note that the "Table Config" tab is only available after you have selected the "Specifics for: Result Summary" or "Specifics for: Parameter Sweep Table" item from the corresponding dropdown menu at the bottom of the dialog box.

The dialog box for the Result Summary is made up out of three tabs:

- One for modulation accuracy results.
- One for power related results.
- One for voltage and current related results. The results in this tab are available after you have activated baseband measurements.

The supported results of the Parameter Sweep Table are part of a single dialog box.

You can add or remove individual results by turning them "On" or "Off".

Remote command:

`DISPlay[:WINDow<n>]:TABLe:ITEM` on page 181

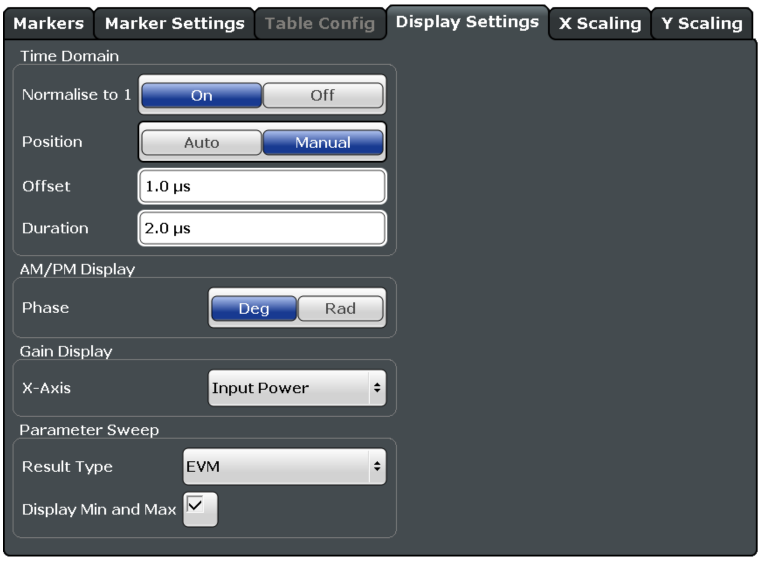
`DISPlay[:WINDow<n>]:PTABle:ITEM` on page 181

4.4 Configuring Result Display Characteristics

The R&S FSW-K18 allows you to define the information displayed in various graphical result displays.

► To access result display characteristics, proceed as follows:

- In the "Configuration Overview", select the "Result Configuration" button and then the "Display Settings" tab.
- Press the MEAS CONFIG key, select the "Result Config" softkey and then the "Display Settings" tab.



Scope of the scaling

The functionality of the "Display Settings" is only available when you have selected one of the result displays that support this feature from the "Specifics for:" dropdown menu at the bottom of the dialog box.

(In this case, the functionality to adjust the Time Domain result display.)

Configuring the Time Domain result display

The "Time Domain" settings select the information displayed in the Time Domain result display and can thus be used to customize the diagram scale.

You can define the characteristics of the x-axis (the amount of displayed data) as well as those of the y-axis (normalized data or actual units).

Available when the Time Domain result display has been selected.

For more information see "Time Domain" on page 20.

Remote command:
Normalization: `DISPlay[:WINDow<n>]:TDOMain:Y[:SCALe]:NORMalise[:STATe]` on page 185
Position: `DISPlay[:WINDow<n>]:TDOMain:X[:SCALe]:MODE` on page 185
Origin: `DISPlay[:WINDow<n>]:TDOMain:X[:SCALe]:OFFSet?` on page 185
Duration: `DISPlay[:WINDow<n>]:TDOMain:X[:SCALe]:DURation?` on page 184

Configuring the AM/PM result display

The "Gain Display" settings select the information displayed in the AM/PM result display.

You can display phase information either in degrees or radians. Select the preferred unit from the corresponding dropdown menu.

Remote command:
`CALCulate<n>:UNIT:ANGLe` on page 183

Configuring the Gain Compression Result Display

The "Gain Display" settings select the information displayed in the Gain Compression result display.

You can analyze the Gain Compression either at the DUT input or at the DUT output. By default, the Gain Compression result display shows the gain against the "Input Power".

To analyze the gain against the output power, select the "Output Power" item from the Gain Display "X-Axis" dropdown menu.

Available when the Gain Compression result display has been selected.

For more information about the Gain Compression result display see "Gain Compression" on page 16.

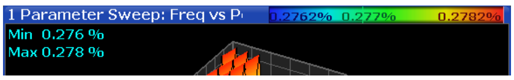
Remote command:
`CALCulate<n>:GAIN:X` on page 183

Selecting the result type displayed in the Parameter Sweep diagram

You can select one of several result types evaluated in the Parameter Sweep diagram. When you open more than one instance of the Parameter Sweep, you can select a different result for each of the instances.

For an extensive list of the supported result types see "Parameter Sweep: Table" on page 23.

By default, the application shows the highest and lowest values that have been measured inside the diagram area.



You can turn that off with the "Display Min and Max" feature.

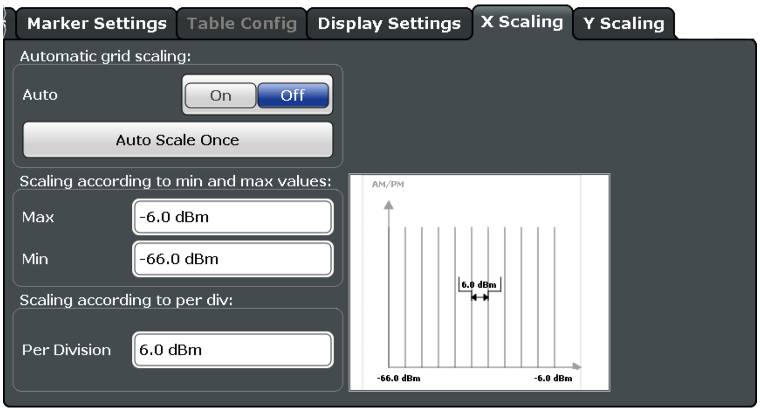
Remote command:
`CONFigure:PSWeep:Z<n>:RESult` on page 183

4.5 Scaling the X-Axis

The R&S FSW-K18 allows you to customize the scale of the x-axis in graphical result displays.

By default, the application automatically scales the x-axis based on the current results. The scale changes when new measurement results are available. When you change the scale manually, the changes are shown in the diagram next to the settings.

► To access scaling functionality, proceed as follows:
- In the "Configuration Overview", select the "Result Configuration" button and then the "X Scaling" tab.
- Press the MEAS CONFIG key, select the "Result Config" softkey and then the "X Scaling" tab..
- Press the AMPT key, and then select the "Scale Config" softkey.



Scope of the scaling

Scaling is applied only to the result display that you have selected from the "Specifics for:" dropdown menu at the bottom of the dialog box.

(In this case, the scale is applied to the AM/PM result display.)

Scaling the x-axis in particular is available for result displays that plot any kind of level values on both axes (for example the AM/PM result display).

Scaling the x-axis automatically

By default, the application scales the x-axis in all diagrams automatically (→ "Auto" = ON).

Automatic scaling tries to obtain the ideal scale for the current measurement results. The application adjusts the scale each time the results change.

You can also force an automatic scaling of the x-axis at any time with the "Auto Scale Once" function. When you select this function, the application scales the x-axis even if the results have not been changed.

Remote command:
`DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:AUTO` on page 186

Scaling the x-axis manually
Settings for manual scaling of the x-axis become available when you turn automatic scaling off.

The application provides two methods to scale the x-axis.

- Scaling according to minimum and maximum values
  The scale is defined by the values at the lower and upper end of the x-axis.
- Scaling according to the distance between two grid lines
  The scale is defined by the value range within two grid lines in the diagram (→ per division). The distance between grid lines refers to diagrams that are split into 10 divisions.

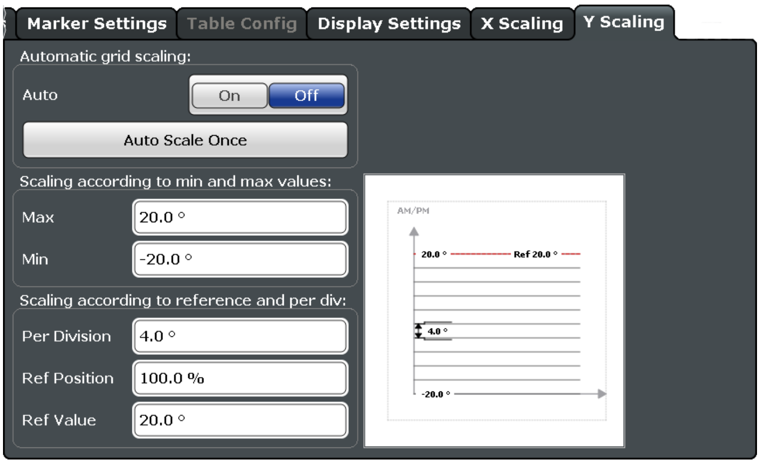
Remote command:
Minimum: `DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:MINimum` on page 187
Maximum: `DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:MAXimum` on page 186
Distance: `DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:PDIVision` on page 187

4.6 Scaling the Y-Axis

The R&S FSW-K18 allows you to customize the scale of the y-axis in graphical result displays.

By default, the application automatically scales the y-axis based on the current results. The scale changes when new measurement results are available. When you change the scale manually, the changes are shown in the diagram next to the settings.

► To access scaling functionality, proceed as follows:
  - In the "Configuration Overview", select the "Result Configuration" button and then the "Y Scaling" tab.
  - Press the MEAS CONFIG key, select the "Result Config" softkey and then the "Y Scaling" tab..
  - Press the AMPT key, and then select the "Scale Config" softkey.



Scope of the scaling

Scaling is applied only to the result display that you have selected from the "Specifics for:" dropdown menu at the bottom of the dialog box.

(In this case, the scale is applied to the Spectrum FFT result display.)

Scaling the y-axis automatically

By default, the application scales the y-axis in all diagrams automatically (→ "Auto" = ON).

Automatic scaling tries to obtain the ideal scale for the current measurement results. The application adjusts the scale each time the results change.

You can also force an automatic scaling of the y-axis at any time with the "Auto Scale Once" function. When you select this function, the application scales the y-axis even if the results have not been changed.

Remote command:
`DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:AUTO` on page 188

Scaling the y-axis manually

Settings for manual scaling of the y-axis become available when you turn automatic scaling off.

The application provides two methods to scale the y-axis.

- Scaling according to minimum and maximum values
  The scale is defined by the values at the lower and upper end of the y-axis.
- Scaling according to reference value
  The scale is defined relative to the reference value and a constant distance between the grid lines (→ per division). The distance between grid lines refers to diagrams that are split into 10 divisions.

The position of the reference value is arbitrary. By default it is at the upper end of the y-axis (100 %).

Remote command:
Minimum: `DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:MAXimum` on page 188
Maximum: `DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:MINimum` on page 189
Reference value: `DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:RVALue` on page 190
Position: `DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:RPOSition` on page 189
Distance: `DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:PDIVision` on page 189

5 Remote Control Commands for Amplifier Measurements

The following remote control commands are required to configure and perform amplifier measurements in a remote environment. The R&S FSW must already be set up for remote operation in a network as described in the base unit manual.

Universal functionality

Note that basic tasks that are also performed in the base unit in the same way are not described here. For a description of such tasks, see the R&S FSW User Manual.

In particular, this includes:

- Managing Settings and Results, i.e. storing and loading settings and result data.
- Basic instrument configuration, e.g. checking the system configuration, customizing the screen layout, or configuring networks and remote operation.
- Using the common status registers (specific status registers for Pulse measurements are not used).

5.1 Overview of Remote Command Suffixes

The remote commands for the LTE Measurement application support the following suffixes.

Suffix	Description
<k>	Selects a limit line. Irrelevant for amplifier measurements.
<m>	Selects a marker.
<n>	Selects a measurement window.
<t>	Selects a trace.

5.2 Introduction

Commands are program messages that a controller (e.g. a PC) sends to the instrument or software. They operate its functions ('setting commands' or 'events') and request information ('query commands'). Some commands can only be used in one way, others work in two ways (setting and query). If not indicated otherwise, the commands can be used for settings and queries.

The syntax of a SCPI command consists of a header and, in most cases, one or more parameters. To use a command as a query, you have to append a question mark after the last header element, even if the command contains a parameter.

A header contains one or more keywords, separated by a colon. Header and parameters are separated by a "white space" (ASCII code 0 to 9, 11 to 32 decimal, e.g. blank). If there is more than one parameter for a command, these are separated by a comma from one another.

Only the most important characteristics that you need to know when working with SCPI commands are described here. For a more complete description, refer to the User Manual of the R&S FSW.

Remote command examples

Note that some remote command examples mentioned in this general introduction may not be supported by this particular application.

5.2.1 Conventions used in Descriptions

Note the following conventions used in the remote command descriptions:

- **Command usage**
  If not specified otherwise, commands can be used both for setting and for querying parameters.
  If a command can be used for setting or querying only, or if it initiates an event, the usage is stated explicitely.
- **Parameter usage**
  If not specified otherwise, a parameter can be used to set a value and it is the result of a query.
  Parameters required only for setting are indicated as **Setting parameters**.
  Parameters required only to refine a query are indicated as **Query parameters**.
  Parameters that are only returned as the result of a query are indicated as **Return values**.
- **Conformity**
  Commands that are taken from the SCPI standard are indicated as **SCPI confirmed**. All commands used by the R&S FSW follow the SCPI syntax rules.
- **Asynchronous commands**
  A command which does not automatically finish executing before the next command starts executing (overlapping command) is indicated as an **Asynchronous command**.
- **Reset values (*RST)**

Default parameter values that are used directly after resetting the instrument (*RST command) are indicated as ***RST** values, if available.

- **Default unit**
  This is the unit used for numeric values if no other unit is provided with the parameter.
- **Manual operation**
  If the result of a remote command can also be achieved in manual operation, a link to the description is inserted.

5.2.2 Long and Short Form

The keywords have a long and a short form. You can use either the long or the short form, but no other abbreviations of the keywords.

The short form is emphasized in upper case letters. Note however, that this emphasis only serves the purpose to distinguish the short from the long form in the manual. For the instrument, the case does not matter.

Example:

`SENSe:FREQuency:CENTer` is the same as `SENS:FREQ:CENT`.

5.2.3 Numeric Suffixes

Some keywords have a numeric suffix if the command can be applied to multiple instances of an object. In that case, the suffix selects a particular instance (e.g. a measurement window).

Numeric suffixes are indicated by angular brackets (<n>) next to the keyword.

If you don't quote a suffix for keywords that support one, a 1 is assumed.

Example:

`DISPlay[:WINDow<1...4>]:ZOOM:STATe` enables the zoom in a particular measurement window, selected by the suffix at `WINDow`.

`DISPlay:WINDow4:ZOOM:STATe ON` refers to window 4.

5.2.4 Optional Keywords

Some keywords are optional and are only part of the syntax because of SCPI compliance. You can include them in the header or not.

Note that if an optional keyword has a numeric suffix and you need to use the suffix, you have to include the optional keyword. Otherwise, the suffix of the missing keyword is assumed to be the value 1.

Optional keywords are emphasized with square brackets.

Example:

Without a numeric suffix in the optional keyword:

`[SENSe:]FREQuency:CENTer` is the same as `FREQuency:CENTer`

With a numeric suffix in the optional keyword:

`DISPlay[:WINDow<1...4>]:ZOOM:STATe`

`DISPlay:ZOOM:STATe ON` enables the zoom in window 1 (no suffix).

`DISPlay:WINDow4:ZOOM:STATe ON` enables the zoom in window 4.

5.2.5 Alternative Keywords

A vertical stroke indicates alternatives for a specific keyword. You can use both keywords to the same effect.

Example:

`[SENSe:]BANDwidth|BWIDth[:RESolution]`

In the short form without optional keywords, `BAND 1MHZ` would have the same effect as `BWID 1MHZ`.

5.2.6 SCPI Parameters

Many commands feature one or more parameters.

If a command supports more than one parameter, these are separated by a comma.

Example:

`LAYout:ADD:WINDow Spectrum,LEFT,MTABle`

Parameters may have different forms of values.

5.2.6.1 Numeric Values

Numeric values can be entered in any form, i.e. with sign, decimal point or exponent. In case of physical quantities, you can also add the unit. If the unit is missing, the command uses the basic unit.

Example:

with unit: `SENSe:FREQuency:CENTer 1GHZ`

without unit: `SENSe:FREQuency:CENTer 1E9` would also set a frequency of 1 GHz.

Values exceeding the resolution of the instrument are rounded up or down.

If the number you have entered is not supported (e.g. in case of discrete steps), the command returns an error.

Instead of a number, you can also set numeric values with a text parameter in special cases.

- MIN/MAX
  Defines the minimum or maximum numeric value that is supported.
- DEF
  Defines the default value.
- UP/DOWN
  Increases or decreases the numeric value by one step. The step size depends on the setting. In some cases you can customize the step size with a corresponding command.

Querying numeric values

When you query numeric values, the system returns a number. In case of physical quantities, it applies the basic unit (e.g. Hz in case of frequencies). The number of digits after the decimal point depends on the type of numeric value.

Example:

Setting: `SENSe:FREQuency:CENTer 1GHZ`

Query: `SENSe:FREQuency:CENTer?` would return `1E9`

In some cases, numeric values may be returned as text.

- INF/NINF
  Infinity or negative infinity. Represents the numeric values 9.9E37 or -9.9E37.
- NAN
  Not a number. Represents the numeric value 9.91E37. NAN is returned in case of errors.

5.2.6.2 Boolean

Boolean parameters represent two states. The "ON" state (logically true) is represented by "ON" or a numeric value 1. The "OFF" state (logically untrue) is represented by "OFF" or the numeric value 0.

Querying boolean parameters

When you query boolean parameters, the system returns either the value 1 ("ON") or the value 0 ("OFF").

Example:

Setting: `DISPlay:WINDow:ZOOM:STATe ON`

Query: `DISPlay:WINDow:ZOOM:STATe?` would return `1`

5.2.6.3 Character Data

Character data follows the syntactic rules of keywords. You can enter text using a short or a long form. For more information see chapter 5.2.2, "Long and Short Form", on page 83.

Querying text parameters

When you query text parameters, the system returns its short form.

Example:

Setting: `SENSe:BANDwidth:RESolution:TYPE NORMal`

Query: `SENSe:BANDwidth:RESolution:TYPE?` would return `NORM`

5.2.6.4 Character Strings

Strings are alphanumeric characters. They have to be in straight quotation marks. You can use a single quotation mark (') or a double quotation mark (").

Example:

`INSTRument:DELete 'Spectrum'`

5.2.6.5 Block Data

Block data is a format which is suitable for the transmission of large amounts of data.

The ASCII character # introduces the data block. The next number indicates how many of the following digits describe the length of the data block. In the example the 4 following digits indicate the length to be 5168 bytes. The data bytes follow. During the transmission of these data bytes all end or other control signs are ignored until all bytes are transmitted. #0 specifies a data block of indefinite length. The use of the indefinite format requires a NL^END message to terminate the data block. This format is useful when the length of the transmission is not known or if speed or other considerations prevent segmentation of the data into blocks of definite length.

5.3 Selecting the Application

INSTrument:CREate:DUPLicate

This command duplicates the currently selected measurement channel, i.e creates a new measurement channel of the same type and with the identical measurement settings. The name of the new channel is the same as the copied channel, extended by a consecutive number (e.g. "IQAnalyzer" -> "IQAnalyzer2").

The channel to be duplicated must be selected first using the `INST:SEL` command.

Example:

```
INST:SEL 'IQAnalyzer'
INST:CRE:DUPL
```

Duplicates the channel named 'IQAnalyzer' and creates a new measurement channel named 'IQAnalyzer2'.

Usage: Event

INSTrument:CREate:REPLace <ChannelName1>,<ChannelType>,<ChannelName2>

This command replaces a measurement channel with another one.

Setting parameters:

<ChannelName1> String containing the name of the measurement channel you want to replace.

<ChannelType> Channel type of the new channel.
For a list of available channel types see `INSTrument:LIST?` on page 88.

<ChannelName2> String containing the name of the new channel.
Note: If the specified name for a new channel already exists, the default name, extended by a sequential number, is used for the new channel (see `INSTrument:LIST?` on page 88).

Example:

```
INST:CRE:REPL 'IQAnalyzer2',IQ,'IQAnalyzer'
```

Replaces the channel named 'IQAnalyzer2' by a new measurement channel of type 'IQ Analyzer' named 'IQAnalyzer'.

Usage: Setting only

INSTrument:CREate[:NEW] <ChannelType>, <ChannelName>

This command adds an additional measurement channel.

The number of measurement channels you can configure at the same time depends on available memory.

Parameters:

<ChannelType> Channel type of the new channel.
For a list of available channel types see `INSTrument:LIST?` on page 88.

<ChannelName> String containing the name of the channel. The channel name is displayed as the tab label for the measurement channel.
Note: If the specified name for a new channel already exists, the default name, extended by a sequential number, is used for the new channel (see `INSTrument:LIST?` on page 88).

Example: `INST:CRE IQ, 'IQAnalyzer2'`
Adds an additional I/Q Analyzer channel named "IQAnalyzer2".

INSTrument:DELete <ChannelName>

This command deletes a measurement channel.

If you delete the last measurement channel, the default "Spectrum" channel is activated.

Parameters:
<ChannelName> String containing the name of the channel you want to delete.
A measurement channel must exist in order to be able delete it.

Example: `INST:DEL 'IQAnalyzer4'`
Deletes the channel with the name 'IQAnalyzer4'.

Usage: Event

INSTrument:LIST?

This command queries all active measurement channels. This is useful in order to obtain the names of the existing measurement channels, which are required in order to replace or delete the channels.

Return values:
<ChannelType>, <ChannelName> For each channel, the command returns the channel type and channel name (see tables below).
Tip: to change the channel name, use the `INSTrument:REName` command.

Example: `INST:LIST?`
Result for 3 measurement channels:
`'ADEM','Analog Demod','IQ','IQ Analyzer','IQ','IQ Analyzer2'`

Usage: Query only

Table 5-1: Available measurement channel types and default channel names in Signal and Spectrum Analyzer mode

Application	<ChannelType> Parameter	Default Channel Name*)
Spectrum	SANALYZER	Spectrum
I/Q Analyzer	IQ	IQ Analyzer
Note: the default channel name is also listed in the table. If the specified name for a new channel already exists, the default name, extended by a sequential number, is used for the new channel.		

Application	<ChannelType> Parameter	Default Channel Name*)
Pulse (R&S FSW-K6)	PULSE	Pulse
Analog Demodulation (R&S FSW-K7)	ADEM	Analog Demod
GSM (R&S FSW-K10)	GSM	GSM
Multi-Carrier Group Delay (R&S FSW-K17)	MCGD	MC Group Delay
Amplifier Measurements (R&S FSW-K18)	AMPLifier	Amplifier
Noise (R&S FSW-K30)	NOISE	Noise
Phase Noise (R&S FSW-K40)	PNOISE	Phase Noise
Transient Analysis (R&S FSW-K60)	TA	Transient Analysis
VSA (R&S FSW-K70)	DDEM	VSA
3GPP FDD BTS (R&S FSW-K72)	BWCD	3G FDD BTS
3GPP FDD UE (R&S FSW-K73)	MWCD	3G FDD UE
TD-SCDMA BTS (R&S FSW-K76)	BTDS	TD-SCDMA BTS
TD-SCDMA UE (R&S FSW-K77)	MTDS	TD-SCDMA UE
cdma2000 BTS (R&S FSW-K82)	BC2K	CDMA2000 BTS
cdma2000 MS (R&S FSW-K83)	MC2K	CDMA2000 MS
1xEV-DO BTS (R&S FSW-K84)	BDO	1xEV-DO BTS
1xEV-DO MS (R&S FSW-K85)	MDO	1xEV-DO MS
WLAN (R&S FSW-K91)	WLAN	WLAN
LTE (R&S FSW-K10x)	LTE	LTE
Real-Time Spectrum (R&S FSW-B160R/-K160RE)	RTIM	Real-Time Spectrum
DOCSIS 3.1 (R&S FSW-K192)	DOCSis	DOCSIS 3.1
Note: the default channel name is also listed in the table. If the specified name for a new channel already exists, the default name, extended by a sequential number, is used for the new channel.		

INSTrument:REName <ChannelName1>, <ChannelName2>

This command renames a measurement channel.

Parameters:

<ChannelName1> String containing the name of the channel you want to rename.

<ChannelName2> String containing the new channel name.
Note that you can not assign an existing channel name to a new channel; this will cause an error.

Example: `INST:REN 'IQAnalyzer2','IQAnalyzer3'`
Renames the channel with the name 'IQAnalyzer2' to 'IQAnalyzer3'.

Usage: Setting only

INSTrument[:SELect] <ChannelType> | <ChannelName>

This command activates a new measurement channel with the defined channel type, or selects an existing measurement channel with the specified name.

Also see

- `INSTrument:CREate[:NEW]` on page 87

Parameters:

<ChannelType> Channel type of the new channel.
For a list of available channel types see `INSTrument:LIST?` on page 88.

<ChannelName> String containing the name of the channel.

Example: `INST IQ`
Activates a measurement channel for the I/Q Analyzer application (evaluation mode).
`INST 'MyIQSpectrum'`
Selects the measurement channel named 'MyIQSpectrum' (for example before executing further commands for that channel).

Usage: SCPI confirmed

SYSTem:PRESet:CHANnel[:EXECute]

This command restores the default instrument settings in the current channel.

Use `INST:SEL` to select the channel.

Example: `INST 'Spectrum2'`
Selects the channel for "Spectrum2".
`SYST:PRES:CHAN:EXEC`
Restores the factory default settings to the "Spectrum2" channel.

Usage: Event

Manual operation: See "Preset Channel" on page 25

5.4 Configuring the Screen Layout

DISPlay:FORMat <Format>

This command determines which tab is displayed.

Parameters:

<Format>

SPLit
Displays the MultiView tab with an overview of all active channels

SINGle
Displays the measurement channel that was previously focused.

*RST: SING

Example: `DISP:FORM SPL`

DISPlay[:WINDow<n>]:SIZE <Size>

This command maximizes the size of the selected result display window *temporarily*. To change the size of several windows on the screen permanently, use the `LAY:SPL` command (see `LAYout:SPLitter` on page 95).

Parameters:

<Size>

LARGe
Maximizes the selected window to full screen.
Other windows are still active in the background.

SMALl
Reduces the size of the selected window to its original size.
If more than one measurement window was displayed originally, these are visible again.

*RST: SMALl

Example: `DISP:WIND2:LARG`

LAYout:ADD[:WINDow]? <WindowName>,<Direction>,<WindowType>

This command adds a window to the display in the active measurement channel.

This command is always used as a query so that you immediately obtain the name of the new window as a result.

To replace an existing window, use the `LAYout:REPLace[:WINDow]` command.

Parameters:

<WindowName> String containing the name of the existing window the new window is inserted next to.
By default, the name of a window is the same as its index. To determine the name and index of all active windows, use the `LAYout:CATalog[:WINDow]?` query.

<Direction> LEFT | RIGHt | ABOVe | BELow
Direction the new window is added relative to the existing window.

<WindowType> text value
Type of result display (evaluation method) you want to add.
See the table below for available parameter values.

Return values:

<NewWindowName> When adding a new window, the command returns its name (by default the same as its number) as a result.

Example: `LAY:ADD? '1',LEFT,MTAB`
Result:
`'2'`
Adds a new window named '2' with a marker table to the left of window 1.

Usage: Query only

Manual operation: See "Numeric Result Summary" on page 10
See "AM/AM" on page 14
See "AM/PM" on page 15
See "Gain Compression" on page 16
See "Magnitude Capture (RF, I and Q)" on page 16
See "PAE vs Input Power / PAE vs Output Power" on page 17
See "PAE vs Time" on page 17
See "Power vs Time" on page 18
See "Raw EVM" on page 18
See "Error Vector Spectrum" on page 19
See "Spectrum FFT" on page 20
See "Time Domain" on page 20
See "Vcc vs Icc" on page 22

Table 5-2: <WindowType> parameter values for Amplifier Measurement application

Parameter value	Window type
ACP	Adjacent Channel Power (Table)
AMAM	AM/AM
AMPM	AM/PM
GCOMpression	Gain Compression
IMAGnitude	Magnitude Capture I
ISPectrum	Spectrum FFT I

Parameter value	Window type
MTABle	Marker Table
PAEI	PAE Input Power
PAEO	PAE Output Power
PAETime	PAE Time
PSWeep	Parameter Sweep (Diagram)
PTABle	Parameter Sweep (Table)
PVTime	Power vs Time (I x Q)
QMAGnitude	Magnitude Capture Q
QSPectrum	Spectrum FFT Q
REVM	Raw EVM
RFMagnitude	Magnitude Capture RF
RFSPectrum	Spectrum FFT
RTABle	Result Summary (Table)
SEVM	Spectrum EVM
TDOMain	Time Domain
VICC	Vcc vs Icc

LAYout:CATalog[:WINDow]?

This command queries the name and index of all active windows in the active measurement channel from top left to bottom right. The result is a comma-separated list of values for each window, with the syntax:

<WindowName_1>,<WindowIndex_1>..<WindowName_n>,<WindowIndex_n>

Return values:

<WindowName> string

Name of the window.
In the default state, the name of the window is its index.

<WindowIndex> **numeric value**
Index of the window.

Example: `LAY:CAT?`
Result:
`'2',2,'1',1`
Two windows are displayed, named '2' (at the top or left), and '1' (at the bottom or right).

Usage: Query only

LAYout:DIRection <Direction>

This command selects the general direction of the smart grid.

Parameters:

<Direction> **HORizontal**

VERTical

*RST: HORizontal

Example: `LAY:DIR HOR`

LAYout:IDENtify[:WINDow]? <WindowName>

This command queries the **index** of a particular display window in the active measurement channel.

Note: to query the **name** of a particular window, use the `LAYout:WINDow<n>:IDENtify?` query.

Query parameters:

<WindowName> String containing the name of a window.

Return values:

<WindowIndex> Index number of the window.

Example: `LAY:WIND:IDEN? '2'`
Queries the index of the result display named '2'.
Response:
`2`

Usage: Query only

LAYout:REMove[:WINDow] <WindowName>

This command removes a window from the display in the active measurement channel.

Parameters:

<WindowName> String containing the name of the window.
In the default state, the name of the window is its index.

Example: `LAY:REM '2'`
Removes the result display in the window named '2'.

Usage: Event

LAYout:REPLace[:WINDow] <WindowName>,<WindowType>

This command replaces the window type (for example from "Diagram" to "Result Summary") of an already existing window in the active measurement channel while keeping its position, index and window name.

To add a new window, use the `LAYout:ADD[:WINDow]?` command.

Parameters:

<WindowName> String containing the name of the existing window.
By default, the name of a window is the same as its index. To determine the name and index of all active windows in the active measurement channel, use the `LAYout:CATalog[:WINDow]?` query.

<WindowType> Type of result display you want to use in the existing window.
See `LAYout:ADD[:WINDow]?` on page 91 for a list of available window types.

Example: `LAY:REPL:WIND '1',MTAB`
Replaces the result display in window 1 with a marker table.

LAYout:SPLitter <Index1>,<Index2>,<Position>

This command changes the position of a splitter and thus controls the size of the windows on each side of the splitter.

Compared to the `DISPlay[:WINDow<n>]:SIZE` on page 91 command, the `LAYout:SPLitter` changes the size of all windows to either side of the splitter permanently, it does not just maximize a single window temporarily.

Note that windows must have a certain minimum size. If the position you define conflicts with the minimum size of any of the affected windows, the command will not work, but does not return an error.

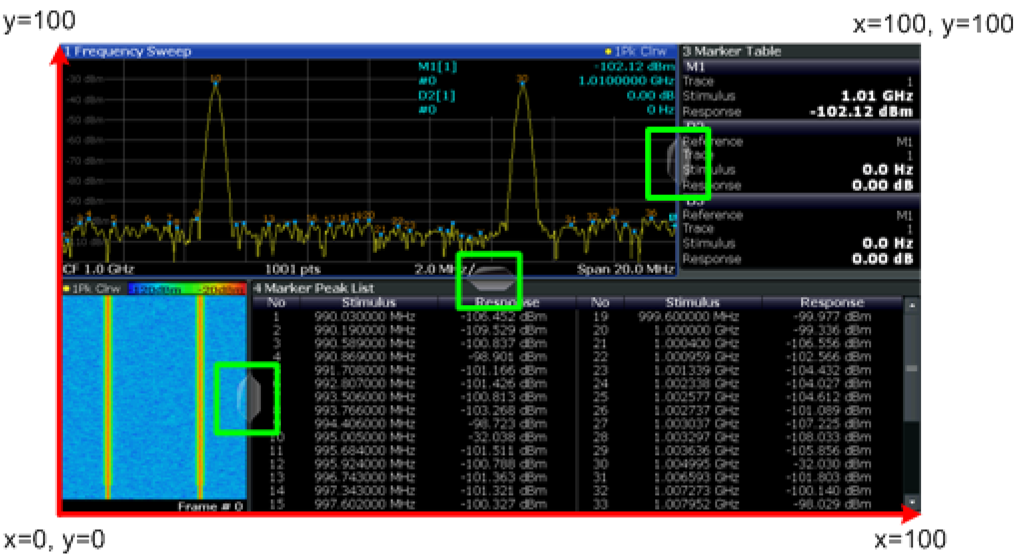


Fig. 5-1: SmartGrid coordinates for remote control of the splitters

Parameters:

<Index1> The index of one window the splitter controls.

<Index2> The index of a window on the other side of the splitter.

<Position>	New vertical or horizontal position of the splitter as a fraction of the screen area (without channel and status bar and softkey menu). The point of origin (x = 0, y = 0) is in the lower left corner of the screen. The end point (x = 100, y = 100) is in the upper right corner of the screen. (See figure 5-1.) The direction in which the splitter is moved depends on the screen layout. If the windows are positioned horizontally, the splitter also moves horizontally. If the windows are positioned vertically, the splitter also moves vertically. Range: 0 to 100
Example:	`LAY:SPL 1,3,50` Moves the splitter between window 1 ('Frequency Sweep') and 3 ('Marker Table') to the center (50%) of the screen, i.e. in the figure above, to the left.

Example:	`LAY:SPL 1,4,70`
	Moves the splitter between window 1 ('Frequency Sweep') and 3 ('Marker Peak List') towards the top (70%) of the screen.
	The following commands have the exact same effect, as any combination of windows above and below the splitter moves the splitter vertically.
	`LAY:SPL 3,2,70`
	`LAY:SPL 4,1,70`
	`LAY:SPL 2,1,70`

LAYout:WINDow<n>:ADD? <Direction>,<WindowType>

This command adds a measurement window to the display. Note that with this command, the suffix <n> determines the existing window next to which the new window is added, as opposed to `LAYout:ADD[:WINDow]?`, for which the existing window is defined by a parameter.

To replace an existing window, use the `LAYout:WINDow<n>:REPLace` command.

This command is always used as a query so that you immediately obtain the name of the new window as a result.

Parameters:

<Direction>	LEFT | RIGHt | ABOVe | BELow
<WindowType>	Type of measurement window you want to add. See `LAYout:ADD[:WINDow]?` on page 91 for a list of available window types.

Return values:

<NewWindowName>	When adding a new window, the command returns its name (by default the same as its number) as a result.

Example: `LAY:WIND1:ADD? LEFT,MTAB`
Result:
`'2'`
Adds a new window named '2' with a marker table to the left of window 1.

Usage: Query only

LAYout:WINDow<n>:IDENtify?

This command queries the **name** of a particular display window (indicated by the <n> suffix) in the active measurement channel.

Note: to query the **index** of a particular window, use the `LAYout:IDENtify[:WINDow]?` command.

Return values:
<WindowName> String containing the name of a window.
In the default state, the name of the window is its index.

Example: `LAY:WIND2:IDEN?`
Queries the name of the result display in window 2.
Response:
`'2'`

Usage: Query only

LAYout:WINDow<n>:REMove

This command removes the window specified by the suffix <n> from the display in the active measurement channel.

The result of this command is identical to the `LAYout:REMove[:WINDow]` command.

Example: `LAY:WIND2:REM`
Removes the result display in window 2.

Usage: Event

LAYout:WINDow<n>:REPLace <WindowType>

This command changes the window type of an existing window (specified by the suffix <n>) in the active measurement channel.

The result of this command is identical to the `LAYout:REPLace[:WINDow]` command.

To add a new window, use the `LAYout:WINDow<n>:ADD?` command.

Parameters:
<WindowType> Type of measurement window you want to replace another one with.
See `LAYout:ADD[:WINDow]?` on page 91 for a list of available window types.

Example: `LAY:WIND2:REPL MTAB`
Replaces the result display in window 2 with a marker table.

LAYout:WINDow<n>:TYPe?

Queries the window type of the window specified by the index <n>. For a list of possible window types see `LAYout:ADD[:WINDow]?` on page 91.

Example: `LAY:WIND2:TYPE?`
Response:
`MACC`
Modulation accuracy

Usage: Query only

5.5 Performing Amplifier Measurements

5.5.1 Performing Measurements

You can include the Amplifier measurements in a sequence of measurements. For a comprehensive description of commands required to do so, please refer to the R&S FSW User Manual.

INITiate<n>:CONMeas

This command restarts a (single) measurement that has been stopped (using `ABORt`) or finished in single measurement mode.

The measurement is restarted at the beginning, not where the previous measurement was stopped.

As opposed to `INITiate<n>[:IMMediate]`, this command does not reset traces in maxhold, minhold or average mode. Therefore it can be used to continue measurements using maxhold or averaging functions.

Suffix:
<n> irrelevant

Usage: Event

Manual operation: See "Continue Single Sweep" on page 27

INITiate<n>:CONTinuous <State>

This command controls the measurement mode for an individual measurement channel.

Note that in single measurement mode, you can synchronize to the end of the measurement with *OPC, *OPC? or *WAI. In continuous measurement mode, synchronization to the end of the measurement is not possible. Thus, it is not recommended that you use continuous measurement mode in remote control, as results like trace data or markers are only valid after a single measurement end synchronization.

If the measurement mode is changed for a measurement channel while the Sequencer is active (see `INITiate<n>:SEQuencer:IMMediate` on page 100) the mode is only considered the next time the measurement in that channel is activated by the Sequencer.

Suffix:
<n> irrelevant

Parameters:
<State> ON | OFF | 0 | 1

ON | 1
Continuous measurement

OFF | 0
Single measurement

*RST: 1

Example:
`INIT:CONT OFF`
Switches the measurement mode to single measurement.
`INIT:CONT ON`
Switches the measurement mode to continuous measurement.

Manual operation: See "Continuous Sweep/RUN CONT" on page 26

INITiate<n>[:IMMediate]

This command starts a (single) new measurement.

You can synchronize to the end of the measurement with *OPC, *OPC? or *WAI.

Suffix:
<n> irrelevant

Usage: Event

Manual operation: See "Single Sweep/ RUN SINGLE" on page 26

INITiate<n>:SEQuencer:ABORt

This command stops the currently active sequence of measurements. The Sequencer itself is not deactivated, so you can start a new sequence immediately using `INITiate<n>:SEQuencer:IMMediate` on page 100.

To deactivate the Sequencer use `SYSTem:SEQuencer` on page 101.

Suffix:

<n> irrelevant

Usage: Event

INITiate<n>:SEQuencer:IMMediate

This command starts a new sequence of measurements by the Sequencer.

Its effect is similar to the `INITiate<n>[:IMMediate]` command used for a single measurement.

Before this command can be executed, the Sequencer must be activated (see `SYSTem:SEQuencer` on page 101).

Suffix:

<n> irrelevant

Example:

`SYST:SEQ ON`
Activates the Sequencer.
`INIT:SEQ:MODE SING`
Sets single sequence mode so each active measurement will be performed once.
`INIT:SEQ:IMM`
Starts the sequential measurements.

Usage: Event

INITiate<n>:SEQuencer:MODE <Mode>

This command selects the way the R&S FSW application performs measurements sequentially.

Before this command can be executed, the Sequencer must be activated (see `SYSTem:SEQuencer` on page 101).

Note: In order to synchronize to the end of a sequential measurement using *OPC, *OPC? or *WAI you must use `SINGle` Sequence mode.

Suffix:

<n> irrelevant

Parameters:

<Mode>

SINGle
Each measurement is performed once (regardless of the channel's sweep mode), considering each channels' sweep count, until all measurements in all active channels have been performed.

CONTinuous
The measurements in each active channel are performed one after the other, repeatedly (regardless of the channel's sweep mode), in the same order, until the Sequencer is stopped.

CDEFined
First, a single sequence is performed. Then, only those channels in continuous sweep mode (`INIT:CONT ON`) are repeated.

*RST: CONTinuous

Example:

`SYST:SEQ ON`
Activates the Sequencer.
`INIT:SEQ:MODE SING`
Sets single sequence mode so each active measurement will be performed once.
`INIT:SEQ:IMM`
Starts the sequential measurements.

SYSTem:SEQuencer <State>

This command turns the Sequencer on and off. The Sequencer must be active before any other Sequencer commands (`INIT:SEQ...`) are executed, otherwise an error will occur.

Parameters:

<State> ON | OFF | 0 | 1

ON | 1
The Sequencer is activated and a sequential measurement is started immediately.

OFF | 0
The Sequencer is deactivated. Any running sequential measurements are stopped. Further Sequencer commands (`INIT:SEQ...`) are not available.

*RST: 0

Example:

`SYST:SEQ ON`
Activates the Sequencer.
`INIT:SEQ:MODE SING`
Sets single Sequencer mode so each active measurement will be performed once.
`INIT:SEQ:IMM`
Starts the sequential measurements.
`SYST:SEQ OFF`

5.5.2 Retrieving Graphical Measurement Results

TRACe<n>[:DATA]? <Trace>

This command queries the measurement results in the graphical result displays. Usually, the measurement results are either displayed on the y-axis (two-dimensional diagrams) or the z-axis (three-dimensional diagrams).

Suffix:

<n> 1..n
Selects the result window.

Query parameters:

<Trace> TRACE1 | ... | TRACE6

Selects the trace to be queried.
Note that the available number of traces depends on the result display.
For example, the Magnitude Capture result display only supports TRACE1, while the Time Domain result display supports TRACE1 to TRACE6.

Return values:

<Result> <numeric value>

Values of the captured samples in chronological order.

Example: `TRAC:DATA TRACE1`
Queries the results displayed on trace 1.

Usage: Query only

Manual operation:
See "AM/AM" on page 14
See "AM/PM" on page 15
See "Gain Compression" on page 16
See "Magnitude Capture (RF, I and Q)" on page 16
See "PAE vs Input Power / PAE vs Output Power" on page 17
See "PAE vs Time" on page 17
See "Power vs Time" on page 18
See "Raw EVM" on page 18
See "Error Vector Spectrum" on page 19
See "Spectrum FFT" on page 20
See "Time Domain" on page 20
See "Vcc vs Icc" on page 22

TRACe<n>[:DATA]:X? <Trace>

This command queries the measurement results as displayed on the x-axis in the graphical result displays.

Suffix:

<n> 1..n
Selects the result window.

Query parameters:

<Trace> TRACE1 | ... | TRACE6

Selects the trace to be queried.
Note that the available number of traces depends on the result display.
For example, the Magnitude Capture result display only supports TRACE1, while the Time Domain result display supports TRACE1 to TRACE6.

Return values:

<Result> <numeric value>

X-axis values of the captured samples in chronological order.

Example: `TRAC:DATA TRACE1`
Queries the results displayed on trace 1.

Usage: Query only

TRACe<n>[:DATA]:Y? <Trace>

This command queries the measurement results as displayed on the y-axis in result displays with three axes (for example the Parameter Sweep).

Suffix:

<n> 1..n
Selects the result window.

Query parameters:

<Trace> TRACE1 | ... | TRACE6

Selects the trace to be queried.
Note that the available number of traces depends on the result display.

Return values:

<Result> <numeric value>

Y-axis values of the captured samples in chronological order.

Example: `TRAC:DATA TRACE1`
Queries the results displayed on trace 1.

Usage: Query only

5.5.3 Retrieving Numeric Results

5.5.3.1 Retrieving General Numeric Results

FETCh:TTF:CURRent[:RESult]?

This command queries the Trigger to Frame result as displayed in the channel bar.

Return values:

<Time> <numeric value>
Default unit: s

Example: `FETC:TTF:CURR?`
would return, e.g.
`0.00015700958`

Usage: Query only

5.5.3.2 Retrieving Results of the Result Summary

Retrieving All Results

FETCh:MACCuracy[:RESult]:ALL?

This command queries all numerical results shown in the Result Summary.

Return values:

<Results> <numerical value>: Results as a comma separated list.
The order of results is the same as in the result summary:
`<RawEVMMin>, <RawEVMCurrent>, <RawEVMMax>, <RawModelEVMMin>, <RawModelEVMCurrent>, <RawModelEVMMax>, ...`
The unit depends on the result.
If a result hasn't been calculated, the command returns `NAN`.

Example: `FETC:MACC:ALL?`
would return, e.g.
`0.277,0.277,0.277,0.002,0.245,0.922,...`

Usage: Query only

Retrieving the Modulation Accuracy

FETCh:MACCuracy:FERRor:MAXimum[:RESult]?
FETCh:MACCuracy:FERRor:MINimum[:RESult]?
FETCh:MACCuracy:FERRor:CURRent[:RESult]?

This command queries the Frequency Error as shown in the Result Summary.

Return values:

<FrequencyError> <numeric value>

Minimum, maximum or current Frequency Error, depending on the command syntax.

Default unit: Hz

Example: `FETC:MACC:FERR:MAX?`
would return, e.g.
`1.2879`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

FETCh:MACCuracy:GIMBalance:MAXimum[:RESult]?
FETCh:MACCuracy:GIMBalance:MINimum[:RESult]?
FETCh:MACCuracy:GIMBalance:CURRent[:RESult]?

This command queries the Gain Imbalance as shown in the Result Summary.

Return values:

<GainImbalance> <numeric value>

Minimum, maximum or current Gain Imbalance, depending on the command syntax.

Default unit: dB

Example: `FETC:MACC:GIMB:MIN?`
would return, e.g.
`0.887`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

FETCh:MACCuracy:IQIMbalance:MAXimum[:RESult]?
FETCh:MACCuracy:IQIMbalance:MINimum[:RESult]?
FETCh:MACCuracy:IQIMbalance:CURRent[:RESult]?

This command queries the I/Q Imbalance as shown in the Result Summary.

Return values:

<IQImbalance> <numeric value>

Minimum, maximum or current I/Q Imbalance, depending on the command syntax.

Default unit: dB

Example: `FETC:MACC:IQIM:CURR?`
would return, e.g.
`0.02`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

FETCh:MACCuracy:IQOFfset:MAXimum[:RESult]?
FETCh:MACCuracy:IQOFfset:MINimum[:RESult]?
FETCh:MACCuracy:IQOFfset:CURRent[:RESult]?

This command queries the I/Q Offset as shown in the Result Summary.

Return values:

<IQOffset> <numeric value>

Minimum, maximum or current I/Q Offset, depending on the command syntax.

Default unit: dB

Example: `FETC:MACC:IQOF:MIN?`
would return, e.g.
`0.001`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

FETCh:MACCuracy:MERRor:MAXimum[:RESult]?
FETCh:MACCuracy:MERRor:MINimum[:RESult]?
FETCh:MACCuracy:MERRor:CURRent[:RESult]?

This command queries the Magnitude Error as shown in the Result Summary.

Return values:

<Magnitude> <numeric value>

Minimum, maximum or current Magnitude Error, depending on the command syntax.

Default unit: %

Example: `FETC:MACC:MERR:MAX?`
would return, e.g.
`1.12`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

FETCh:MACCuracy:PERRor:MAXimum[:RESult]?
FETCh:MACCuracy:PERRor:MINimum[:RESult]?
FETCh:MACCuracy:PERRor:CURRent[:RESult]?

This command queries the Phase Error as shown in the Result Summary.

Return values:

<PhaseError> <numeric value>

Minimum, maximum or current Phase Error, depending on the command syntax.

Default unit: degree

Example: `FETC:MACC:PERR:CURR?`
would return, e.g.
`1.84`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

FETCh:MACCuracy:QERRor:MAXimum[:RESult]?
FETCh:MACCuracy:QERRor:MINimum[:RESult]?
FETCh:MACCuracy:QERRor:CURRent[:RESult]?

This command queries the Quadrature Error as shown in the Result Summary.

Return values:

<QuadratureError> <numeric value>

Minimum, maximum or current Quadrature Error, depending on the command syntax.

Default unit: degree

Example: `FETC:MACC:QERR:MAX?`
would return, e.g.
`2.76`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

FETCh:MACCuracy:REVM:MAXimum[:RESult]?
FETCh:MACCuracy:REVM:MINimum[:RESult]?
FETCh:MACCuracy:REVM:CURRent[:RESult]?

This command queries the Raw EVM as shown in the Result Summary.

Return values:

<EVM> <numeric value>

Minimum, maximum or current Raw EVM, depending on the command syntax.

Default unit: %

Example: `FETC:MACC:REVM:MAX?`
would return, e.g.
`3.606`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

FETCh:MACCuracy:RMEV:MAXimum[:RESult]?
FETCh:MACCuracy:RMEV:MINimum[:RESult]?
FETCh:MACCuracy:RMEV:CURRent[:RESult]?

This command queries the Raw Model EVM as shown in the Result Summary.

Return values:

<EVM> <numeric value>

Minimum, maximum or current Raw Model EVM, depending on the command syntax.

Default unit: %

Example: `FETC:MACC:RMEV:CURR?`
would return, e.g.
`0.879`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

FETCh:MACCuracy:SRERror:MAXimum[:RESult]?
FETCh:MACCuracy:SRERror:MINimum[:RESult]?
FETCh:MACCuracy:SRERror:CURRent[:RESult]?

This command queries the Sample Rate Error as shown in the Result Summary.

Return values:

<SampleRateError> <numeric value>

Minimum, maximum or current SampleRateError, depending on the command syntax.

Default unit: Hz

Example: `FETC:MACC:SRER:CURR?`
would return, e.g.
`-0.023`

Usage: Query only

Manual operation: See "Results to check modulation accuracy" on page 11

Retrieving Power Results

FETCh:AMAM:CWIDth:CURRent[:RESult]?

This command queries the AM/AM Curve Width as shown in the Result Summary.

Return values:

<CurveWidth> <numeric value>

Current AM/AM Curve Width.

Default unit: dB

Example: `FETC:AMAM:CWID:CURR?`
would return, e.g.
`0.69`

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

FETCh:AMPM:CWIDth:CURRent[:RESult]?

This command queries the AM/PM Curve Width as shown in the Result Summary.

Return values:
<CurveWidth> <numeric value>
Current AM/PM Curve Width.
Default unit: degree

Example: `FETC:AMPM:CWID:CURR?`
would return, e.g.
`1.441`

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

FETCh:POWer:CFACtor:IN:CURRent[:RESult]?

This command queries the Crest Factor at the DUT input as shown in the Result Summary.

Return values:
<CrestFactor> <numeric value>
Current Crest Factor.
Default unit: dB

Example: `FETC:POW:CFAC:IN:CURR?`
would return, e.g.
`10.34`

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

FETCh:POWer:CFACtor:OUT:CURRent[:RESult]?

This command queries the Crest Factor at the DUT output as shown in the Result Summary.

Return values:
<CrestFactor> <numeric value>
Current Crest Factor.
Default unit: dB

Example: `FETC:POW:CFAC:CURR?`
would return, e.g.
`8.72`

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

FETCh:POWer:GAIN:MAXimum[:RESult]?
FETCh:POWer:GAIN:MINimum[:RESult]?
FETCh:POWer:GAIN:CURRent[:RESult]?

This command queries the signal gain as shown in the Result Summary.

Return values:

<Gain> <numeric value>

Minimum, maximum or current gain, depending on the command syntax.

Default unit: dB

Example:
```
FETC:POW:GAIN:MAX?
```
would return, e.g.
```
21.37
```

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

FETCh:POWer:INPut:MAXimum[:RESult]?
FETCh:POWer:INPut:MINimum[:RESult]?
FETCh:POWer:INPut:CURRent[:RESult]?

This command queries the power at the DUT input as shown in the Result Summary.

Return values:

<Power> <numeric value>

Minimum, maximum or current power, depending on the command syntax.

Default unit: dBm

Example:
```
FETC:POW:INP:MIN?
```
would return, e.g.
```
9.39
```

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

FETCh:POWer:OUTPut:MAXimum[:RESult]?
FETCh:POWer:OUTPut:MINimum[:RESult]?
FETCh:POWer:OUTPut:CURRent[:RESult]?

This command queries the signal power at the DUT output as shown in the Result Summary.

Return values:

<Power> — <numeric value>

Minimum, maximum or current power, depending on the command syntax.

Default unit: dBm

Example: `FETC:POW:OUTP:MIN?`
would return, e.g.
`7.198`

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

FETCh:POWer:P1DB:CURRent[:RESult]?

This command queries the 1 dB Compression Point as shown in the Result Summary.

Return values:

<Level> — <numeric value>

Current 1 dB Compression Point.

Default unit: dBm

Example: `FETC:POW:P1DB:CURR?`
would return, e.g.
`-5.782`

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

FETCh:POWer:P2DB:CURRent[:RESult]?

This command queries the 2 dB Compression Point as shown in the Result Summary.

Return values:

<Level> — <numeric value>

Current 2 dB Compression Point.

Default unit: dBm

Example: `FETC:POW:P2DB:CURR?`
would return, e.g.
`-8.193`

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

FETCh:POWer:P3DB:CURRent[:RESult]?

This command queries the 3 dB Compression Point as shown in the Result Summary.

Return values:

<Level> <numeric value>

Current 3 dB Compression Point.

Default unit: dBm

Example: `FETC:POW:P3DB:CURR?`
would return, e.g.
`2.551`

Usage: Query only

Manual operation: See "Results to check power characteristcs" on page 12

Retrieving Baseband Characteristics

FETCh:APAE:MAXimum[:RESult]?
FETCh:APAE:MINimum[:RESult]?
FETCh:APAE:CURRent[:RESult]?

This command queries the Average PAE (Power Added Efficiency) as shown in the Result Summary.

Return values:

<PAE> <numeric value>

Minimum, maximum or current Average PAE, depending on the command syntax.

Default unit: %

Example: `FETC:APAE:CURR?`
would return, e.g.
`1.231`

Usage: Query only

Manual operation: See "Results to check the power supply characteristics of the amplifier" on page 12

FETCh:BBPower:MAXimum[:RESult]?
FETCh:BBPower:MINimum[:RESult]?
FETCh:BBPower:CURRent[:RESult]?

This command queries the measured baseband power (I_cc * V_cc) as shown in the Result Summary.

Return values:

>Power> <numeric value>

Minimum, maximum or current power, depending on the command syntax.

Default unit: W

Example: `FETC:BBP:CURR?`
would return, e.g.
`0.75`

Usage: Query only

Manual operation: See "Results to check the power supply characteristics of the amplifier" on page 12

FETCh:ICC:MAXimum[:RESult]?
FETCh:ICC:MINimum[:RESult]?
FETCh:ICC:CURRent[:RESult]?

This command queries the measured baseband current (I_cc) as shown in the Result Summary.

Return values:

<Current> Minimum, maximum or current current, depending on the command syntax.

Default unit: A

Example: `FETC:ICC:MAX?`
would return, e.g.
`2.63`

Usage: Query only

Manual operation: See "Results to check the power supply characteristics of the amplifier" on page 12

FETCh:IVOLtage:PURE:MAXimum[:RESult]?
FETCh:IVOLtage:PURE:MINimum[:RESult]?
FETCh:IVOLtage:PURE:CURRent[:RESult]?

This command queries the voltage measured at the baseband input I as shown in the Result Summary.

The returned value is a "pure" voltage that does not contain any correction factors.

Return values:

<Voltage> <numeric value>

Minimum, maximum or current voltage, depending on the command syntax.

Default unit: V

Example: `FETC:IVOL:PURE:CURR?`
would return, e.g.
`1.4`

Usage: Query only

Manual operation: See "Results to check the power supply characteristics of the amplifier" on page 12

FETCh:QVOLtage:PURE:MAXimum[:RESult]?
FETCh:QVOLtage:PURE:MINimum[:RESult]?
FETCh:QVOLtage:PURE:CURRent[:RESult]?

This command queries the measured at the baseband input Q as shown in the Result Summary.

The returned value is a "pure" voltage that does not contain any correction factors.

Return values:

<Voltage> <numeric value>

Minimum, maximum or current voltage, depending on the command syntax.

Default unit: V

Example: `FETC:IVOL:PURE:CURR?`
would return, e.g.
`1.42`

Usage: Query only

Manual operation: See "Results to check the power supply characteristics of the amplifier" on page 12

FETCh:VCC:MAXimum[:RESult]?
FETCh:VCC:MINimum[:RESult]?
FETCh:VCC:CURRent[:RESult]?

This command queries the measured baseband voltage (V_cc) as shown in the Result Summary.

Return values:

<Current> Minimum, maximum or current current, depending on the command syntax.

Default unit: V

Example: `FETC:VCC:CURR?`
would return, e.g.
`0.4`

Usage: Query only

Manual operation: See "Results to check the power supply characteristics of the amplifier" on page 12

5.5.3.3 Retrieving Results of the Parameter Sweep Table

Retrieving the results in the Parameter Sweep Table requires six commands for every result type.

Example command set to query the EVM results:

- `FETCh:PTABle:EVM:MAXimum[:RESult]` queries the highest EVM that has been measured.
- `FETCh:PTABle:EVM:MAXimum:X[:RESult]` queries the location on the x-axis where the highest EVM has been measured.
- `FETCh:PTABle:EVM:MAXimum:Y[:RESult]` queries the location on the y-axis where the highest EVM has been measured.
- `FETCh:PTABle:EVM:MINimum[:RESult]` queries the lowest EVM that has been measured.
- `FETCh:PTABle:EVM:MINimum:X[:RESult]` queries the location on the x-axis where the lowest EVM has been measured.
- `FETCh:PTABle:EVM:MINimum:Y[:RESult]` queries the location on the y-axis where the lowest EVM has been measured.

The type and unit of the value queried on the x- and y-axes depends on the parameter you have selected with `CONFigure:PSWeep:X:SETTing` and `CONFigure:PSWeep:Y:SETTing`.

FETCh:PTABle[:RESult]:ALL?

This command queries all numerical results shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>: Results as a comma separated list.
`<EVMMinValue>, <EVMMinX>, <EVMMinY>, <ACPMinCalue>, <ACPMinX>, <ACPMinY>,...`
The unit depends on the result and parameters assigned to the x- and y-axis.
If a result hasn't been calculated, the command returns `NAN`.

Example: `FETC:PTAB:ALL?`
would return, e.g.
`0.244445,1e+007,-30,0.246109,2e+007,-30,-21.9096,3e+007,-3`
`[etc.]`

Usage: Query only

FETCh:PTABle:ACP:MAXimum:X[:RESult]
FETCh:PTABle:ACP:MAXimum:Y[:RESult]
FETCh:PTABle:ACP:MAXimum[:RESult]
FETCh:PTABle:ACP:MINimum:X[:RESult]
FETCh:PTABle:ACP:MINimum:Y[:RESult]
FETCh:PTABle:ACP:MINimum[:RESult]?

These commands query the result values for the ACP result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:ACP:MAX?`
would return, e.g.
`-7.651[DBM]`

Usage: Query only

FETCh:PTABle:ACP:ACHannel<n>:LOWer:MAXimum:X[:RESult]
FETCh:PTABle:ACP:ACHannel<n>:LOWer:MAXimum:Y[:RESult]
FETCh:PTABle:ACP:ACHannel<n>:LOWer:MAXimum[:RESult]
FETCh:PTABle:ACP:ACHannel<n>:LOWer:MINimum:X[:RESult]
FETCh:PTABle:ACP:ACHannel<n>:LOWer:MINimum:Y[:RESult]
FETCh:PTABle:ACP:ACHannel<n>:LOWer:MINimum[:RESult]?

These commands query the result values for the lower adjacent channel power as shown in the Parameter Sweep Table.

Suffix:

<n> irrelevant

Return values:

<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:ACP:ACH:LOW:MIN?`
would return, e.g.
`-10.945[DBM]`

Usage: Query only

FETCh:PTABle:ACP:AChannel<n>:UPPer:MAXimum:X[:RESult]
FETCh:PTABle:ACP:AChannel<n>:UPPer:MAXimum:Y[:RESult]
FETCh:PTABle:ACP:AChannel<n>:UPPer:MAXimum[:RESult]
FETCh:PTABle:ACP:AChannel<n>:UPPer:MINimum:X[:RESult]
FETCh:PTABle:ACP:AChannel<n>:UPPer:MINimum:Y[:RESult]
FETCh:PTABle:ACP:AChannel<n>:UPPer:MINimum[:RESult]?

These commands query the result values for the upper adjacent channel power as shown in the Parameter Sweep Table.

Suffix:
<n> irrelevant

Return values:
<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:ACP:ACH:UPP:MIN:Y?`
would return, e.g. (if the y-axis represents the frequency)
`10000000[HZ]`

Usage: Query only

FETCh:PTABle:AMAM:CWIDth:MAXimum:X[:RESult]
FETCh:PTABle:AMAM:CWIDth:MAXimum:Y[:RESult]
FETCh:PTABle:AMAM:CWIDth:MAXimum[:RESult]
FETCh:PTABle:AMAM:CWIDth:MINimum:X[:RESult]
FETCh:PTABle:AMAM:CWIDth:MINimum:Y[:RESult]
FETCh:PTABle:AMAM:CWIDth:MINimum[:RESult]?

These commands query the result values for the AM/AM Curve Width result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:AMAM:CWID:MAX:X?`
would return, e.g. (if the x-axis represents the RF to envelope delay)
`-0.000001[s]`

Usage: Query only

FETCh:PTABle:AMPM:CWIDth:MAXimum:X[:RESult]
FETCh:PTABle:AMPM:CWIDth:MAXimum:Y[:RESult]
FETCh:PTABle:AMPM:CWIDth:MAXimum[:RESult]
FETCh:PTABle:AMPM:CWIDth:MINimum:X[:RESult]
FETCh:PTABle:AMPM:CWIDth:MINimum:Y[:RESult]
FETCh:PTABle:AMPM:CWIDth:MINimum[:RESult]?

These commands query the result values for the AM/PM Curve Width result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:AMPM:CWID:MAX:X?`
would return, e.g. (if the x-axis represents the frequency)
`150000000[HZ]`

Usage: Query only

FETCh:PTABle:BBPower:MAXimum:X[:RESult]
FETCh:PTABle:BBPower:MAXimum:Y[:RESult]
FETCh:PTABle:BBPower:MAXimum[:RESult]
FETCh:PTABle:BBPower:MINimum:X[:RESult]
FETCh:PTABle:BBPower:MINimum:Y[:RESult]
FETCh:PTABle:BBPower:MINimum[:RESult]?

These commands query the result values for the Baseband Power (I_cc * V_cc) result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:VCC:MIN:Y?`
would return, e.g. (if the y-axis represents the envelope bias)
`-0.10000000149[V]`

Usage: Query only

FETCh:PTABle:CFACtor:MAXimum:X[:RESult]
FETCh:PTABle:CFACtor:MAXimum:Y[:RESult]
FETCh:PTABle:CFACtor:MAXimum[:RESult]
FETCh:PTABle:CFACtor:MINimum:X[:RESult]
FETCh:PTABle:CFACtor:MINimum:Y[:RESult]
FETCh:PTABle:CFACtor:MINimum[:RESult]?

These commands query the result values for the Crest Factor result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:CFAC:MIN?`
would return, e.g.
`0.053[DB]`

Usage: Query only

FETCh:PTABle:EVM:MAXimum:X[:RESult]
FETCh:PTABle:EVM:MAXimum:Y[:RESult]
FETCh:PTABle:EVM:MAXimum[:RESult]
FETCh:PTABle:EVM:MINimum:X[:RESult]
FETCh:PTABle:EVM:MINimum:Y[:RESult]
FETCh:PTABle:EVM:MINimum[:RESult]?

These commands query the result values for the EVM result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:EVM:MAX:Y?`
would return, e.g. (if the y-axis represents the output power)
`0[DBM]`

Usage: Query only

FETCh:PTABle:GAIN:MAXimum:X[:RESult]
FETCh:PTABle:GAIN:MAXimum:Y[:RESult]
FETCh:PTABle:GAIN:MAXimum[:RESult]
FETCh:PTABle:GAIN:MINimum:X[:RESult]
FETCh:PTABle:GAIN:MINimum:Y[:RESult]
FETCh:PTABle:GAIN:MINimum[:RESult]?

These commands query the result values for the Gain result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For ...[:RESult]: Minimum or maximum result that has been measured.
• For ...:X[:RESult]: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (CONFigure:PSWeep:X:SETTing).
• For ...:Y[:RESult]: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (CONFigure:PSWeep:Y:SETTing).

Example: FETC:PTAB:GAIN:MAX?
would return, e.g.
-5.392[DBM]

Usage: Query only

FETCh:PTABle:ICC:MAXimum:X[:RESult]
FETCh:PTABle:ICC:MAXimum:Y[:RESult]
FETCh:PTABle:ICC:MAXimum[:RESult]
FETCh:PTABle:ICC:MINimum:X[:RESult]
FETCh:PTABle:ICC:MINimum:Y[:RESult]
FETCh:PTABle:ICC:MINimum[:RESult]?

These commands query the result values for the I_cc result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For ...[:RESult]: Minimum or maximum result that has been measured.
• For ...:X[:RESult]: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (CONFigure:PSWeep:X:SETTing).
• For ...:Y[:RESult]: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (CONFigure:PSWeep:Y:SETTing).

Example: FETC:PTAB:VCC:MIN:Y?
would return, e.g. (if the y-axis represents the output power)
-10[DBM]

Usage: Query only

FETCh:PTABle:PAE:MAXimum:X[:RESult]
FETCh:PTABle:PAE:MAXimum:Y[:RESult]

FETCh:PTABle:PAE:MAXimum[:RESult]
FETCh:PTABle:PAE:MINimum:X[:RESult]
FETCh:PTABle:PAE:MINimum:Y[:RESult]
FETCh:PTABle:PAE:MINimum[:RESult]?

These commands query the result values for the PAE result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:PAE:MAX?`
would return, e.g.
`89.3[PCT]`

Usage: Query only

FETCh:PTABle:RMS:MAXimum:X[:RESult]
FETCh:PTABle:RMS:MAXimum:Y[:RESult]
FETCh:PTABle:RMS:MAXimum[:RESult]
FETCh:PTABle:RMS:MINimum:X[:RESult]
FETCh:PTABle:RMS:MINimum:Y[:RESult]
FETCh:PTABle:RMS:MINimum[:RESult]?

These commands query the result values for the RMS Power result as shown in the Parameter Sweep Table.

Return values:

<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:RMS:MIN?`
would return, e.g.
`-12.032[DBM]`

Usage: Query only

FETCh:PTABle:VCC:MAXimum:X[:RESult]
FETCh:PTABle:VCC:MAXimum:Y[:RESult]
FETCh:PTABle:VCC:MAXimum[:RESult]
FETCh:PTABle:VCC:MINimum:X[:RESult]
FETCh:PTABle:VCC:MINimum:Y[:RESult]
FETCh:PTABle:VCC:MINimum[:RESult]?

These commands query the result values for the V_cc result as shown in the Parameter Sweep Table.

Return values:
<Results> <numeric value>

• For `...[:RESult]`: Minimum or maximum result that has been measured.
• For `...:X[:RESult]`: Location on the x-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the x-axis (`CONFigure:PSWeep:X:SETTing`).
• For `...:Y[:RESult]`: Location on the y-axis where the minimum or maximum result has been measured. The type of value depends on the parameter you have selected for the y-axis (`CONFigure:PSWeep:Y:SETTing`).

Example: `FETC:PTAB:VCC:MIN:X?`
would return, e.g. (if the x-axis represents the frequency)
`10000000[HZ]`

Usage: Query only

5.6 Configuring Amplifier Measurements

5.6.1 Designing a Reference Signal

CONFigure:REFSignal:CGW:LEDState?

This command queries the transmission state of the reference signal designed on a signal generator.

Available when you configure the reference signal on a signal generator.

Return values:

<State>

GREen
Transmission was successful.

GREY
Unknown transmission state.

RED
Transmission was not successful.

Example:

```
CONF:REFS:CGW:READ
CONF:REFS:CGW:LEDS?
```

would return, e.g.

```
GRE
```

Usage: Query only

Manual operation: See "Designing a reference signal on a signal generator" on page 29

CONFigure:REFSignal:CGW:READ

This command transfers a reference signal designed on a signal generator into the R&S FSW-K18.

Example: `CONF:REFS:CGW:READ`
Imports the reference signal data from the generator.

Usage: Event

Manual operation: See "Designing a reference signal on a signal generator" on page 29

CONFigure:REFSignal:CWF:DPIPower <Power>

This command defines the peak input power of the DUT.

This is necessary when you turn off `CONFigure:REFSignal:CWF:ETGenerator[:STATe]`.

Parameters:
<Power> <numeric value>

Default unit: dBm

Example: `CONF:REFS:CWF:ETG OFF`
`CONF:REFS:CWF:DPIP 3`
Defines a DUT input power of 3 dBm.

Manual operation: See "Designing a reference signal in a waveform file" on page 30

CONFigure:REFSignal:CWF:ETGenerator[:STATe] <State>

This command turns the transfer of the reference signal data to a generator on and off.

Parameters:
<State> **ON**
Reference signal data is transferred to the generator and generated with the generator.

OFF
Reference signal data is not transferred to the generator but simulated by the analyzer based on the information of the waveform file.
When you turn it off, you have to define the peak input power of the DUT with `CONFigure:REFSignal:CWF:DPIPower`.

*RST: ON

Example: `CONF:REFS:CWF:ETG OFF`
Analyzer simulates the reference signal.

Manual operation: See "Designing a reference signal in a waveform file" on page 30

CONFigure:REFSignal:CWF:FPATh <FileName>

This command selects a waveform file containing a reference signal.

Parameters:

<FileName> String containing the name and path to the waveform file.

Example:
`CONF:REFS:CWF:FPAT 'C:\RefSignal.wv'`
Selects a waveform file on drive c: called 'RefSignal.wv'.

Manual operation: See "Designing a reference signal in a waveform file" on page 30

CONFigure:REFSignal:CWF:LEDState?

This command queries the transmission state of the reference signal to the signal generator.

Available when you generate the reference signal with a waveform file.

Return values:

<State>

GREen
Transmission was successful.

GREY
Unknown transmission state.

RED
Transmission was not successful.

Example:
```
CONF:REFS:CWF:FPAT 'C:\RefSignal.wv'
CONF:REFS:CWF:WRITE
CONF:REFS:CWF:LEDS?
```
would return, e.g.
`GRE`

Usage: Query only

Manual operation: See "Designing a reference signal in a waveform file" on page 30

CONFigure:REFSignal:CWF:WRITe

This command transfers a waveform file that contains a reference signal to a signal generator.

Example:
```
CONF:REFS:CWF:FPAT 'C:\RefSignal.wv'
CONF:REFS:CWF:WRITE
```
Transfers the reference signal to the generator.

Usage: Event

Manual operation: See "Designing a reference signal in a waveform file" on page 30

CONFigure:REFSignal:GOS:BWIDth <Bandwidth>

This command defines the bandwidth of the reference signal.

Parameters:

<Bandwidth> <numeric value>

Default unit: Hz

Example: `CONF:REFS:GOS:BWID 10MHZ`

Defines a reference signal bandwidth of 10 MHz.

Manual operation: See "Signal Bandwidth" on page 32

CONFigure:REFSignal:GOS:CRESt <CrestFactor>

This command defines the crest factor of the reference signal.

Parameters:

<CrestFactor> <numeric value>

Default unit: dB

Example: `CONF:REFS:GOS:CRES 15`

Defines a crest factor of 15 dB.

Manual operation: See "Crest Factor" on page 33

CONFigure:REFSignal:GOS:DCYCle <DutyCycle>

This command defines the duty cycle of a pulsed reference signal.

Parameters:

<DutyCycle> <numeric value>

Default unit: %

Example: `CONF:REFS:GOS:DCYC 75`

Defines a duty cycle of 75 %.

Manual operation: See "Pulse Duty Cycle" on page 33

CONFigure:REFSignal:GOS:LEDState?

This command queries the transmission state of the reference signal to the signal generator.

Available when you configure the reference signal within the R&S FSW-K18.

Return values:

<State>

GREen
Transmission was successful.

GREY
Unknown transmission state.

RED
Transmission was not successful.

Example:
`CONF:REFS:GOS:WRITE`
`CONF:REFS:GOS:LEDS?`
would return, e.g.
`GRE`

Usage: Query only

Manual operation: See "Designing a reference signal within the R&S FSW-K18" on page 31

CONFigure:REFSignal:GOS:NPOSition <Frequency>

This command defines the offset of a notch relative to the center frequency in the reference signal.

Parameters:

<Frequency> <numeric value>

Default unit: Hz

Example:
`CONF:REFS:GOS:NPOS 10000`
Defines a notch offset of 10 kHz.

Manual operation: See "Notch Position" on page 33

CONFigure:REFSignal:GOS:NWIDth <Frequency>

This command defines the notch width of a reference signal.

Parameters:

<Frequency> <numeric value>

Default unit: Hz

Example:
`CONF:REFS:GOS:NWID 150000`
Defines a notch width of 150 kHz.

Manual operation: See "Notch Width" on page 33

CONFigure:REFSignal:GOS:RLENgth <Samples>

This command defines the ramp length of a pulsed reference signal.

Parameters:

<Samples>	<numeric value>: (integer only) Number of samples on each side of the pulse (= ramp length). Default unit: Samples
Example:	`CONF:REFS:GOS:RLEN 5` Defines a ramp length of 5 samples.
Manual operation:	See "Ramp Length" on page 34

CONFigure:REFSignal:GOS:SLENgth <Samples>

This command defines the length of the reference signal.

Parameters:

<Samples>	<numeric value>: (integer only) Default unit: Samples
Example:	`CONF:REFS:GOS:SLEN 1024` Defines a reference signal made up out of 1024 samples.
Manual operation:	See "Signal Length" on page 33

CONFigure:REFSignal:GOS:SRATe <SampleRate>

This command defines the clock (or sample) rate of the reference signal generated by the signal generator.

Parameters:

<SampleRate>	<numeric value> Default unit: Hz
Example:	`CONF:REFS:GOS:SRAT 20000000` Defines a sample rate of 20 MHz.
Manual operation:	See "Clock Rate" on page 32

CONFigure:REFSignal:GOS:WNAMe <FileName>

This command defines a file name for the waveform of the reference signal.

Parameters:

<FileName>	String containing the name of the waveform file. The file extension (.wv) is added automatically.
Example:	`CONF:REFS:GOS:WNAM 'RefSignal'` Defines the name "RefSignal" for the waveform file.
Manual operation:	See "Waveform File Name" on page 34

CONFigure:REFSignal:GOS:WRITe

This command transfers the reference signal characteristics defined within the R&S FSW-K18 to a signal generator.

Example: `CONF:REFS:GOS:WRITE`
Transfers the reference signal to the generator.

Usage: Event

Manual operation: See "Designing a reference signal within the R&S FSW-K18" on page 31

CONFigure:REFSignal:SEGMent <Segment>

This command selects the segment in a multi-waveform file that should be used to generate the reference signal.

Parameters:

<Segment> <numeric value>: (integer only)

Range: Depends on the number of segments in the waveform file.

*RST: 0

Example: `CONF:REFS:SEGM 3`
Selects the 3rd segment in the waveform file.

CONFigure:REFSignal:SINFo:SLENgth?

This command queries the sample length of the currently used reference signal.

Return values:

<Samples> <numeric value>: (integer only)

Default unit: Samples

Example: `CONF:REFS:SINF:SLEN?`
would return, e.g.
`40000`

Usage: Query only

CONFigure:REFSignal:SINFo:SRATe?

This command queries the sample rate of the currently used reference signal.

Return values:

<SampleRate> <numeric value>

Default unit: Hz

Example: `CONF:REFS:SINF:SRAT?`
would return, e.g.
`32000000`

Usage: Query only

5.6.2 Selecting and Configuring the Input Source

CALibration:AIQ:HATiming[:STATe] <State>

Activates a mode with enhanced timing accuracy between analog baseband, RF and external trigger signals.

Parameters:

<State> ON | OFF | 1 | 0

ON | 1
The high accuracy timing function is switched on.
The cable for high accuracy timing must be connected to trigger ports 1 and 2.

OFF | 0
The high accuracy timing function is switched off.

*RST: OFF

Example: `CAL:AIQ:HAT:STAT ON`

Manual operation: See "High Accuracy Timing Trigger - Baseband - RF" on page 38

INPut:CONNector <ConnType>

Determines whether the RF input data is taken from the RF input connector or the optional Analog Baseband I connector. This command is only available if the Analog Baseband interface (R&S FSW-B71) is installed and active for input. It is not available for the R&S FSW67 or R&S FSW85.

For more information on the Analog Baseband Interface (R&S FSW-B71) see the R&S FSW I/Q Analyzer and I/Q Input User Manual.

Parameters:

<ConnType>

RF
RF input connector

AIQI
Analog Baseband I connector

*RST: RF

Example: `INP:CONN:AIQI`
Selects the analog baseband input.

Usage: SCPI confirmed

Manual operation: See "Input Connector" on page 37

INPut:COUPling <CouplingType>

This command selects the coupling type of the RF input.

Parameters:
<CouplingType>

AC
AC coupling

DC
DC coupling

*RST: AC

Example: `INP:COUP DC`

Usage: SCPI confirmed

Manual operation: See "Input Coupling" on page 35

INPut:DPATh <State>

Enables or disables the use of the direct path for frequencies close to 0 Hz.

Parameters:
<State>

AUTO | 1
(Default) the direct path is used automatically for frequencies close to 0 Hz.

OFF | 0
The analog mixer path is always used.

*RST: 1

Example: `INP:DPAT OFF`

Usage: SCPI confirmed

Manual operation: See "Direct Path" on page 36

INPut:FILTer:HPASs[:STATe] <State>

Activates an additional internal high-pass filter for RF input signals from 1 GHz to 3 GHz. This filter is used to remove the harmonics of the R&S FSW in order to measure the harmonics for a DUT, for example.

This function requires an additional high-pass filter hardware option.

(Note: for RF input signals outside the specified range, the high-pass filter has no effect. For signals with a frequency of approximately 4 GHz upwards, the harmonics are suppressed sufficiently by the YIG filter.)

Parameters:

<State>	ON | OFF	
	*RST:	OFF

Example: `INP:FILT:HPAS ON`
Turns on the filter.

Usage: SCPI confirmed

Manual operation: See "High-Pass Filter 1...3 GHz" on page 36

INPut:FILTer:YIG[:STATe] <State>

This command turns the YIG-preselector on and off.

Note the special conditions and restrictions for the YIG filter described in "YIG-Preselector" on page 36.

Parameters:

<State>	ON | OFF | 0 | 1	
	*RST:	1 (0 for I/Q Analyzer, GSM, VSA and MC Group Delay measurements)

Example: `INP:FILT:YIG OFF`
Deactivates the YIG-preselector.

Manual operation: See "YIG-Preselector" on page 36

INPut:IMPedance <Impedance>

This command selects the nominal input impedance of the RF input. In some applications, only 50 Ω are supported.

75 Ω should be selected if the 50 Ω input impedance is transformed to a higher impedance using a matching pad of the RAZ type (= 25 Ω in series to the input impedance of the instrument). The power loss correction value in this case is 1.76 dB = 10 log (75Ω/50Ω).

Parameters:

<Impedance>	50 | 75	
	*RST:	50 Ω

Example: `INP:IMP 75`

Usage: SCPI confirmed

Manual operation: See "Impedance" on page 36

INPut:IQ:BALanced[:STATe] <State>

This command defines whether the input is provided as a differential signal via all 4 Analog Baseband connectors or as a plain I/Q signal via 2 single-ended lines.

Parameters:

<State>	**ON** Differential **OFF** Single ended *RST: ON
Example:	`INP:IQ:BAL OFF`
Manual operation:	See "Input Configuration" on page 38

INPut:SELect:BBANalog[:STATe] <State>

This command turns simultaneous use of RF input and analog baseband input on and off.

Parameters:

<State>	ON | OFF
Example:	`INP:SEL:BBAN ON` Turns the analog baseband on.
Manual operation:	See "Enable Parallel BB Capture" on page 38

5.6.3 Configuring the Frequency

[SENSe:]FREQuency:CENTer <Frequency>

This command defines the center frequency.

Parameters:

<Frequency>	The allowed range and f_{max} is specified in the data sheet. **UP** Increases the center frequency by the step defined using the `[SENSe:]FREQuency:CENTer:STEP` command. **DOWN** Decreases the center frequency by the step defined using the `[SENSe:]FREQuency:CENTer:STEP` command. *RST: fmax/2 Default unit: Hz

Example:	`FREQ:CENT 100 MHz`
	`FREQ:CENT:STEP 10 MHz`
	`FREQ:CENT UP`
	Sets the center frequency to 110 MHz.

Usage:	SCPI confirmed

Manual operation: See "Center Frequency" on page 39

[SENSe:]FREQuency:CENTer:STEP <StepSize>

This command defines the center frequency step size.

Parameters:

<StepSize> f_{max} is specified in the data sheet.

Range: 1 to fMAX
*RST: 0.1 x span
Default unit: Hz

Example:
```
FREQ:CENT 100 MHz
FREQ:CENT:STEP 10 MHz
FREQ:CENT UP
```
Sets the center frequency to 110 MHz.

Manual operation: See "Center Frequency Stepsize" on page 39

[SENSe:]FREQuency:OFFSet <Offset>

This command defines a frequency offset.

If this value is not 0 Hz, the application assumes that the input signal was frequency shifted outside the application. All results of type "frequency" will be corrected for this shift numerically by the application.

Parameters:

<Offset> Range: -100 GHz to 100 GHz
*RST: 0 Hz

Example: `FREQ:OFFS 1GHZ`

Usage: SCPI confirmed

Manual operation: See "Frequency Offset" on page 39

5.6.4 Defining Level Characteristics

DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:RLEVel <ReferenceLevel>

This command defines the reference level (for all traces, <t> is irrelevant).

Parameters:

<ReferenceLevel> The unit is variable.

Range: see datasheet
*RST: 0 dBm

Example: `DISP:TRAC:Y:RLEV -60dBm`

Usage: SCPI confirmed

Manual operation: See "Reference Level" on page 41

DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:RLEVel:OFFSet <Offset>

This command defines a reference level offset (for all traces, <t> is irrelevant).

Parameters:

<Offset> Range: -200 dB to 200 dB
*RST: 0dB

Example: `DISP:TRAC:Y:RLEV:OFFS -10dB`

Manual operation: See "Shifting the Display (Offset)" on page 41

INPut:ATTenuation <Attenuation>

This command defines the total attenuation for RF input.

If you set the attenuation manually, it is no longer coupled to the reference level, but the reference level is coupled to the attenuation. Thus, if the current reference level is not compatible with an attenuation that has been set manually, the command also adjusts the reference level.

Parameters:

<Attenuation> Range: see data sheet
Increment: 5 dB
*RST: 10 dB (AUTO is set to ON)

Example: `INP:ATT 30dB`
Defines a 30 dB attenuation and decouples the attenuation from the reference level.

Usage: SCPI confirmed

Manual operation: See "Attenuation Mode / Value" on page 42

INPut:ATTenuation:AUTO <State>

This command couples or decouples the attenuation to the reference level. Thus, when the reference level is changed, the R&S FSW determines the signal level for optimal internal data processing and sets the required attenuation accordingly.

Parameters:
<State> ON | OFF | 0 | 1

*RST: 1

Example: `INP:ATT:AUTO ON`
Couples the attenuation to the reference level.

Usage: SCPI confirmed

Manual operation: See "Attenuation Mode / Value" on page 42

INPut:EATT <Attenuation>

This command defines an electronic attenuation manually. Automatic mode must be switched off (`INP:EATT:AUTO OFF`, see `INPut:EATT:AUTO` on page 140).

If the current reference level is not compatible with an attenuation that has been set manually, the command also adjusts the reference level.

Parameters:
<Attenuation> attenuation in dB

Range: see data sheet
Increment: 1 dB
*RST: 0 dB (OFF)

Example: `INP:EATT:AUTO OFF`
`INP:EATT 10 dB`

Manual operation: See "Using Electronic Attenuation" on page 43

INPut:EATT:AUTO <State>

This command turns automatic selection of the electronic attenuation on and off.

If on, electronic attenuation reduces the mechanical attenuation whenever possible.

Parameters:
<State> ON | OFF | 0 | 1

*RST: 1

Example: `INP:EATT:AUTO OFF`

Manual operation: See "Using Electronic Attenuation" on page 43

INPut:EATT:STATe <State>

This command turns the electronic attenuator on and off.

Parameters:
<State> ON | OFF

*RST: OFF

Example: `INP:EATT:STAT ON`
Switches the electronic attenuator into the signal path.

Manual operation: See "Using Electronic Attenuation" on page 43

INPut:IQ:FULLscale:LEVel <PeakVoltage>

This command defines the peak voltage to be displayed in the diagram.

The range of the power scale is then defined by +<PeakVoltage> to -<PeakVoltage>.

Parameters:

<PeakVoltage> Peak voltage level

*RST: 1Vp

Example: `INP:IQ:FULL 3V`
Selects a peak voltage of 3 V.

Manual operation: See "Full Scale Level" on page 41

INPut:GAIN[:VALue] <Gain>

This command selects the gain level if the preamplifier is activated (`INP:GAIN:STAT ON`, see `INPut:GAIN:STATe` on page 141).

The command requires the additional preamplifier hardware option.

Parameters:

<Gain> 15 dB | 30 dB

The availability of gain levels depends on the model of the R&S FSW.
R&S FSW8/13: 15dB and 30 dB
R&S FSW26 or higher: 30 dB
All other values are rounded to the nearest of these two.

*RST: OFF

Example: `INP:GAIN:VAL 30`
Switches on 30 dB preamplification.

Usage: SCPI confirmed

Manual operation: See "Preamplifier" on page 42

INPut:GAIN:STATe <State>

This command turns the preamplifier on and off. It requires the optional preamplifiier hardware.

Parameters:

<State> ON | OFF

*RST: OFF

Example: `INP:GAIN:STAT ON`
Switches on 30 dB preamplification.

Usage: SCPI confirmed

Manual operation: See "Preamplifier" on page 42

5.6.5 Controlling a Signal Generator

CONFigure:GENerator:EXTernal:ROSCillator <Source>

This command selects the source of the generator reference frequency.

Parameters:

<Source>

EXT
The generator uses the reference frequency of the analyzer.

INT
The generator uses its own reference frequency.

Example:
`CONF:GEN:EXT:ROSC INT`
Selects the reference frequency of the generator.

CONFigure:GENerator:EXTernal:ROSCillator:LEDState?

This command queries the state of the connection to the external reference.

Return values:

<State>

GREen
Connection was successful.

GREY
Unknown connection state.

RED
Connection was not successful.

Example:
`CONF:GEN:EXT:ROSC:LEDS?`
would return, e.g.:
`RED`

Usage: Query only

CONFigure:GENerator:FREQuency:CENTer <Frequency>

This command defines the frequency of the generator.

Parameters:

<Frequency> <numeric value>

Default unit: Hz

Example: `CONF:GEN:FREQ:CENT 100000000`

Defines a generator frequency of 100 MHz.

Manual operation: See "Center Frequency" on page 46

CONFigure:GENerator:FREQuency:CENTer:LEDState?

This command queries the frequency synchronization state.

Return values:

<State>

GREen
Frequency synchronization was successful.

GREY
Unknown frequency synchronization state.

RED
Frequency synchronization was not successful.

Example: `CONF:GEN:FREQ:CENT:LEDS?`

would return, e.g.:

`GRE`

Usage: Query only

Manual operation: See "Center Frequency" on page 46

CONFigure:GENerator:FREQuency:CENTer:SYNC[:STATe] <State>

This command turns synchronization of the analyzer and generator frequency on and off.

Parameters:

<State> ON | OFF

Example: `CONF:GEN:FREQ:CENT:SYNC ON`

Synchronizes the analyzer and generator frequency when you change it.

Manual operation: See "Attach to R&S FSW Frequency" on page 46

CONFigure:GENerator:IPConnection:ADDRess <IPAddress>

This command defines the IP address of the connected signal generator.

Parameters:

<IPAddress> String containing the IP address.

Example: `CONF:GEN:IPC:ADDR '192.0.2.0'`
Connects the generator with the stated IP address.

Manual operation: See "Generator IP Address" on page 45

CONFigure:GENerator:IPConnection:LEDState?

This command queries the state of connection to the signal generator.

Return values:

<State>

GREen
Connection was successful.

GREY
Unknown connection state.

RED
Connection was not successful.

Example: `CONF:GEN:IPC:LEDS?`
would return, e.g.:
`RED`

Usage: Query only

Manual operation: See "Generator IP Address" on page 45

CONFigure:GENerator:POWer:LEVel <Level>

This command defines the signal generator RMS level.

Parameters:

<Level> <numeric value>
Default unit: dBm

Example: `CONF:GEN:POW:LEV 0`
Defines a level of 0 dBm.

Manual operation: See "Generator RMS Level" on page 45

CONFigure:GENerator:POWer:LEVel:LEDState?

This command queries the level configuration state on the generator.

Return values:

<State>

GREen
Level configuration was successful.

GREY
Unknown level configuration state.

RED
Level configuration was not successful.

Example: `CONF:GEN:POW:LEV:LEDS?`
would return, e.g.:
`GRE`

Usage: Query only

Manual operation: See "Generator RMS Level" on page 45

CONFigure:GENerator:POWer:LEVel:OFFSet <Level>

This command defines a mathematical level offset for the signal generator (for example to take external attenuation into account).

Parameters:

<Level> <numeric value>

Default unit: dBm

Example: `CONF:GEN:POW:LEV:OFFS 10`
Defines a level offset of 10 dBm.

Manual operation: See "Generator RMS Level" on page 45

CONFigure:GENerator:POWer:LEVel:OFFSet:LEDState?

This command queries the level offset configuration state on the generator.

Return values:

<State>

GREen
Level offset configuration was successful.

GREY
Unknown level offset configuration state.

RED
Level offset configuration was not successful.

Example: `CONF:GEN:POW:LEV:LEDS?`
would return, e.g.:
`GRE`

Usage: Query only

Manual operation: See "Generator RMS Level" on page 45

CONFigure:GENerator:SEGMent <Segment>

This command selects the segment in a multi-waveform file that should be uploaded to the signal generator.

Parameters:

<Segment> <numeric value>: (integer only)

Range: Depends on the number of segments in the waveform file.

*RST: 0

Example: `CONF:GEN:SEGM 3`
Selects the 3rd segment of a waveform file.

Manual operation: See "Selecting a segment in a multi segment waveform file" on page 46

CONFigure:GENerator:SEGMent:LEDState?

This command queries the state of transmission of a multi-waveform segment.

Return values:

<State>

GREen
Transmission was successful.

GREY
Unknown transmission state.

RED
Transmission was not successful.

Example: `CONF:GEN:SEGM:LEDS?`
would return, e.g.
`RED`

Usage: Query only

Manual operation: See "Selecting a segment in a multi segment waveform file" on page 46

CONFigure:GENerator:SETTings:UPDate

This command updates the generator settings as defined within the R&S FSW-K18.

Usage: Event

CONFigure:GENerator:TARGet:PATH:BB?

This command queries the signal path of the R&S SMW used for baseband signal generation.

Note that the baseband path is always the same as the RF path selected with `CONFigure:GENerator:TARGet:PATH:RF`.

Return values:

<Path> A | B

Example: `CONF:GEN:TARG:PATH:BB?`
would return, e.g.
`A`

Usage: Query only

Manual operation: See "Path RF / BB" on page 46

CONFigure:GENerator:TARGet:PATH:RF <Path>

This command selects the signal path of the R&S SMW used for RF signal generation.

Parameters:

<Path> A | B

Example: `CONF:GEN:TARG:PATH:RF A`
Selects RF path A to generate the signal.

Manual operation: See "Path RF / BB" on page 46

5.6.6 Configuring the Data Capture

[SENSe:][REFSig:]TIME?

This command queries the length of the reference signal as shown in the "Acquisition" dialog box.

Return values:

<Duration> <numeric value>

Default unit: s

Example: `REFS:TIME?`
would return, e.g.:
`0.00125`

Usage: Query only

Manual operation: See "Configuring the measurement time" on page 49

[SENSe:]SWAPiq <State>

This command inverts the I and Q branches of the signal.

Parameters:

<State> ON | OFF

Example: `SWAP ON`
Inverts the I and Q channel.

Manual operation: See "Inverting the I/Q branches" on page 50

[SENSe:]SWEep:LENGth <Samples>

This command defines the capture length.

This command is available when `[SENSe:]SWEep:TIME:AUTO` has been turned off.

Note that when you change the capture length, the capture time is adjusted automatically to the new capture length.

Parameters:

<Samples> <numeric value>: (integer only)

Default unit: Samples

Example:
```
SWE:TIME:AUTO OFF
SWE:LENG 1000000
```
Defines a capture length of 1 million samples.

Manual operation: See "Configuring the measurement time" on page 49

[SENSe:]SWEep:TIME <Time>

This command defines the capture time.

This command is available when `[SENSe:]SWEep:TIME:AUTO` has been turned off.

Note that when you change the capture time, the capture length is adjusted automatically to the new capture time.

Parameters:

<Time> <numeric value>

Default unit: s

Example:
```
SWE:TIME:AUTO OFF
SWE:TIME 10MS
```
Defines a sweep time of 10 ms.

Manual operation: See "Configuring the measurement time" on page 49

[SENSe:]SWEep:TIME:AUTO <State>

This command turns automatic selection of an appropriate capture time on and off.

When you turn this feature on, the application calculates an appropriate capture time based on the reference signal and adjusts the other acquisition settings accordingly.

Parameters:

<State> ON | OFF

*RST: ON

Example:
```
SWE:TIME:AUTO ON
```
Selects automatic adjustment of the capture time.

Manual operation: See "Configuring the measurement time" on page 49

TRACe:IQ:BWIDth <Bandwidth>

This command defines the analysis bandwidth with which the amplified signal is captured.

This command is available when `TRACe:IQ:SRATe:AUTO` has been turned off.

Note that when you change the analysis bandwidth, the sample rate and capture length are adjusted automatically to the new bandwidth.

Parameters:

<Bandwidth> <numeric value>

Note that the application automatically adjust the sample rate when you change the bandwidth manually.

Default unit: Hz

Example:

```
TRAC:IQ:SRAT:AUTO OFF
TRAC:IQ:BWID 50MHZ
```

Defines a bandwidth of 50 MHz. The sample rate is adjusted accordingly.

Manual operation: See "Configuring the measurement bandwidth" on page 48

TRACe:IQ:SRATe <SampleRate>

This command defines the sample rate with which the amplified signal is captured.

This command is available when `TRACe:IQ:SRATe:AUTO` has been turned off.

Note that when you change the sample rate, the analysis bandwidth and capture length are adjusted automatically to the new sample rate.

Parameters:

<SampleRate> <numeric value>

Note that the application automatically adjust the analysis bandwidth when you change the sample rate manually.

Default unit: Hz

Example:

```
TRAC:IQ:SRAT:AUTO OFF
TRAC:IQ:SRAT 20MHZ
```

Defines a sample rate of 20 MHz. The analysis bandwidth is adjusted accordingly.

Manual operation: See "Configuring the measurement bandwidth" on page 48

TRACe:IQ:SRATe:AUTO <State>

This command turns automatic selection of an appropriate (capture) sample rate on and off.

When you turn this feature on, the application calculates an appropriate sample rate based on the reference signal and adjusts the other data acquisition settings accordingly.

Parameters:

<State> ON | OFF

*RST: ON

Example: `TRAC:IQ:SRAT:AUTO ON`
Selects automatic adjustment of the sample rate.

Manual operation: See "Configuring the measurement bandwidth" on page 48

TRACe:IQ:WBANd:MBWidth <Bandwidth>

This command selects the largest possible bandwidth that can be applied for the wide-band signal path.

The wideband signal path is available with the corresponding bandwidth extensions available for the R&S FSW (for example R&S FSW-B160).

The command is available when you turn on `TRACe:IQ:WBANd[:STATe]`.

Parameters:

<Bandwidth>

80MHZ
Restricts the bandwidth to 80 MHz.
(The wideband signal path is not used in that case. `TRACe:IQ:WBANd[:STATe]` is turned off.)

160MHZ | 320MHZ | 500MHZ
Restricts the bandwidth to the corresponding value, even if you have installed a higher bandwidth extension.

Default unit: Hz

Example: `TRAC:IQ:WBAN ON`
`TRAC:IQ:WBAN:MBW 160MHZ`
Restricts the bandwidth to 160 MHz.

Manual operation: See "Configuring the measurement bandwidth" on page 48

TRACe:IQ:WBANd[:STATe] <State>

This command turns the wideband signal path on and off.

The wideband signal path is available with the corresponding bandwidth extensions available for the R&S FSW (for example R&S FSW-B160).

Parameters:

<State>

ON
Turns on the wideband signal path.
By default, the application allows you to use the maximum available bandwidth ("Auto" mode in manual operation).
You have to turn on the wideband signal path when you want to use bandwidths greater than 80 MHz.

OFF
Turns off the wideband signal path. The largest available bandwidth is 80 MHz.

Example: `TRAC:IQ:WBAN OFF`
Turns off the wideband signal path.

Manual operation: See "Configuring the measurement bandwidth" on page 48

5.6.7 Synchronizing Measurement Data

CONFigure:ESTimation:FULL <State>

This command turns estimation over the complete reference signal on and off.

Parameters:
<State> ON | OFF
When you turn estimation over the full reference signal off, you can define a estimation range with:
- `CONFigure:ESTimation:STARt`
- `CONFigure:ESTimation:STOP`

*RST: ON

Example:
```
CONF:EST:FULL OFF
CONF:EST:STAR 0s
CONF:EST:STOP 20us
```
Defines a synchronization range over the first 20 µs of the capture buffer.

Manual operation: See "Defining the estimation range" on page 52

CONFigure:ESTimation:STARt <Start>

This command defines the start value of the estimation range.

Parameters:
<Start> <numeric value>
Default unit: s

Example: See `CONFigure:ESTimation:FULL`.

Manual operation: See "Defining the estimation range" on page 52

CONFigure:ESTimation:STOP <Stop>

This command defines the end value of the estimation range.

Parameters:

<Stop>	<numeric value> Default unit: s
Example:	See `CONFigure:ESTimation:FULL`.
Manual operation:	See "Defining the estimation range" on page 52

CONFigure:SYNC:CONFidence <Confidence>

This command defines the synhronization confidence level.

Parameters:

<Confidence>	<numeric value> Range: 0 to 100 Default unit: PCT
Example:	`CONF:SYNC:CONF 99` Defines a confidence level of 99 %.
Manual operation:	See "Defining a synchronization confidence level" on page 52

CONFigure:SYNC:DOMain <Domain>

This command selects the synchronization method.

Parameters:

<Domain>	**IQDirect** I/Q data for the reference signal is directly correlated with the reference and measured signal. **IQPDiff** Correlation on the phase differentiated I/Q data. **MAGNitude** Correlation on the magnitude of the I/Q data with no regard for phase information. **TRIGger** It is assumed that the capture is triggered at the start of the reference waveform.
Example:	`CONF:SYNC:DOM IQD` Tries to find a correlation in the raw I/Q data.
Manual operation:	See "Selecting the synchronization method" on page 51

CONFigure:SYNC:SOFail <State>

This command turns a measurement stop when synchronization of measured and reference signal fails on and off.

Parameters:

<State>	ON | OFF *RST: OFF
Example:	`CONF:SYNC:SOF ON` Stops the measurement when synchronization fails.
Manual operation:	See "Turning synchronization of reference and measured signal on and off" on page 51

CONFigure:SYNC:STATe <State>

This command turns synchronization between reference and measured signal on and off.

Parameters:

<State>	ON | OFF *RST: ON
Example:	`CONF:SYNC:STAT ON` Turns on synchronization between reference and measured signal.
Manual operation:	See "Turning synchronization of reference and measured signal on and off" on page 51

FETCh:SYNC:FAIL?

This command queries the synchronization state.

Return values:

<State>	**ON** Synchronization was successful. **OFF** Synchronization was not successful.

Example:	`FETC:SYNC:FAIL?`
	would return, e.g.
	`OFF`

Usage:	Query only

5.6.8 Defining the Evaluation Range

CONFigure:EVALuation:FULL <State>

This command turns result evaluation over the complete capture buffer on and off.

Parameters:
<State> ON | OFF

When you turn calculation over the full capture buffer off, you can define an evaluation range with:
- CONFigure:EVALuation:STARt
- CONFigure:EVALuation:STOP

*RST: ON

Example:
```
CONF:EVAL:FULL OFF
CONF:EVAL:STAR 5us
CONF:EVAL:STOP 50us
```
Defines an evaluation range over 45 µs of the capture buffer.

Manual operation: See "Defining the evaluation range" on page 53

CONFigure:EVALuation:STARt <EvaluationStart>

This command defines the start value of the evaluation range.

Parameters:
<EvaluationStart> <numeric value>

Default unit: s

Example: See CONFigure:EVALuation:FULL.

Manual operation: See "Defining the evaluation range" on page 53

CONFigure:EVALuation:STOP <EvaluationStop>

This command defines the end value of the evaluation range.

Parameters:
<EvaluationStop> <numeric value>

Default unit: s

Example: See CONFigure:EVALuation:FULL.

Manual operation: See "Defining the evaluation range" on page 53

5.6.9 Estimating and Compensating Signal Errors

CONFigure:SIGNal:ERRor:COMPensation:ADRoop[:STATe] <State>

This command turns compensation of the Amplitude Droop on and off.

Available when you turn on `CONFigure:SIGNal:ERRor:ESTimation:ADRoop[:STATe]`.

Parameters:

<State> ON | OFF

*RST: ON

Example: `CONF:SIGN:ERR:COMP:ADR ON`
Turns on error compensation.

CONFigure:SIGNal:ERRor:COMPensation:FOFFset:[STATe] <State>

This command turns compensation of the Frequency Offset on and off.

Available when you turn on `CONFigure:SIGNal:ERRor:ESTimation:FOFFset:[STATe]`.

Parameters:

<State> ON | OFF

*RST: ON

Example: `CONF:SIGN:ERR:COMP:FOFF ON`
Turns on error compensation.

CONFigure:SIGNal:ERRor:COMPensation:IQIMbalance[:STATe] <State>

This command turns compensation of the I/Q Imbalance on and off.

Available when you turn on `CONFigure:SIGNal:ERRor:ESTimation:IQIMbalance[:STATe]`.

Parameters:

<State> ON | OFF

*RST: ON

Example: `CONF:SIGN:ERR:COMP:IQIM ON`
Turns on error compensation.

CONFigure:SIGNal:ERRor:COMPensation:IQOFfset:[STATe] <State>

This command turns compensation of the Sample Error Rate on and off.

Available when you turn on `CONFigure:SIGNal:ERRor:ESTimation:IQOFfset:[STATe]`.

Parameters:

<State> ON | OFF

*RST: ON

Example: `CONF:SIGN:ERR:COMP:IQOF ON`
Turns on error compensation.

CONFigure:SIGNal:ERRor:COMPensation:SRATe[:STATe] <State>

This command turns compensation of the Sample Error Rate on and off.

Available when you turn on `CONFigure:SIGNal:ERRor:ESTimation:SRATe[:STATe]`.

Parameters:
<State> ON | OFF
*RST: ON

Example: `CONF:SIGN:ERR:COMP:SRAT ON`
Turns on error compensation.

CONFigure:SIGNal:ERRor:ESTimation:ADRoop[:STATe] <State>

This command turns estimation of the Amplitude Droop on and off.

Parameters:
<State> ON | OFF
*RST: ON

Example: `CONF:SIGN:ERR:EST:ADR ON`
Turns on error estimation.

CONFigure:SIGNal:ERRor:ESTimation:FOFFset:[STATe] <State>

This command turns estimation of the Frequency Offset on and off.

Parameters:
<State> ON | OFF
*RST: ON

Example: `CONF:SIGN:ERR:EST:FOFF ON`
Turns on error estimation.

CONFigure:SIGNal:ERRor:ESTimation:IQIMbalance[:STATe] <State>

This command turns estimation of the I/Q Imbalance on and off.

Parameters:
<State> ON | OFF
*RST: ON

Example: `CONF:SIGN:ERR:EST:IQIM ON`
Turns on error estimation.

CONFigure:SIGNal:ERRor:ESTimation:IQOFfset:[STATe] <State>

This command turns estimation of the I/Q Offset on and off.

Parameters:

<State> ON | OFF

*RST: ON

Example: `CONF:SIGN:ERR:EST:IQOF ON`
Turns on error estimation.

CONFigure:SIGNal:ERRor:ESTimation:SRATe[:STATe] <State>

This command turns estimation of the Sample Error Rate on and off.

Parameters:

<State> ON | OFF

*RST: ON

Example: `CONF:SIGN:ERR:EST:SRAT ON`
Turns on error estimation.

5.6.10 Applying a System Model

CONFigure:MODeling:AMAM:ORDer <Order>

This command defines the order (or degree) of the AM/AM model polynomials that should be calculated.

Parameters:

<Order> String containing the polynomials to be calculated.
You can either select a range of polynomials (e.g. "1-7"), a selection of polynomials (e.g. "1,3,5") or a combination of both (e.g. "1,3-5").

Range: 0 to 18
*RST: "1-7"

Example: `CONF:MOD:AMAM:ORD "1-5"`
Calculates the polynomials to the 1st, 2nd, 3rd, 4th and 5th degree.

Example: `CONF:MOD:AMAM:ORD "1,3,5"`
Calculates the polynomials to the 1st, 3rd and 5th degree.

Manual operation: See "Selecting the degree of the polynomial" on page 56

CONFigure:MODeling:AMPM:ORDer <Order>

This command defines the order (or degree) of the AM/PM model polynomials that should be calculated.

Parameters:

<Order> String containing the polynomials to be calculated.
You can either select a range of polynomials (e.g. "1-7"), a selection of polynomials (e.g. "1,3,5") or a combination of both (e.g. "1,3-5").

Range: 0 to 18
*RST: "1-7"

Example: `CONF:MOD:AMPM:ORD "1,3-5"`
Calculates the polynomials to the 1st, 3rd, 4th and 5th degree.

Manual operation: See "Selecting the degree of the polynomial" on page 56

CONFigure:MODeling:LRANge <Level>

This command defines the modeling level range.

Parameters:

<Level> <numeric value>

Default unit: dB

Example: `CONF:MOD:LRAN 30`
Defines a modeling level range of 30 dB.

Manual operation: See "Defining the modeling range" on page 56

CONFigure:MODeling:NPOints <Points>

This command defines the number of modeling points.

Parameters:

<Points> <numeric value>: (integer only)

*RST: 50
Default unit: ---

Example: `CONF:MOD:NPO 100`
Calculates the model based on 50 points.

Manual operation: See "Defining the modeling range" on page 56

CONFigure:MODeling:SEQuence <State>

This command selects the sequence in which the models are calculated.

Parameters:
<State> **AMFirst**
Calculates the AM/AM model before calculating the AM/PM model.
PMFirst
Calculates the AM/PM model before calculating the AM/AM model.
*RST: AMFirst

Example: `CONF:MOD:SEQ AMF`
Calculates the AM/AM model first.

Manual operation: See "Turning system modeling on and off" on page 55

CONFigure:MODeling[:STATe] <State>

This command turns system modeling on and off.

Parameters:
<State> ON | OFF
*RST: OFF

Example: `CONF:MOD ON`
Turns on system modeling.

Manual operation: See "Turning system modeling on and off" on page 55

5.6.11 Applying Digital Predistortion

CONFigure:DPD:AMAM[:STATe] <State>

This command turns calculation of AM/AM predistortion on and off.

Parameters:
<State> ON | OFF
*RST: ON

Example: `CONF:DPD:AMAM ON`
Turns on calculation of AM/AM curve.

Manual operation: See "Selecting the order of model calculation" on page 58

CONFigure:DPD:AMPM[:STATe] <State>

This command turns calculation of AM/PM predistortion on and off.

Parameters:

<State> ON | OFF

*RST: ON

Example: `CONF:DPD:AMPM ON`

Turns on calculation of AM/PM curve.

Manual operation: See "Selecting the order of model calculation" on page 58

CONFigure:DPD:AMXM[:STATe] <State>

This command turns calculation of AM/AM and AM/PM predistortion on and off (at the same time).

Alternatively, you can do that with

- `CONFigure:DPD:AMAM[:STATe]`
  and
- `CONFigure:DPD:AMPM[:STATe]`

Parameters:

<State> ON | OFF

Example: `CONF:DPD:AMXM ON`

Calculates both AM/AM and AM/PM predistortion.

Usage: Setting only

Manual operation: See "Selecting the order of model calculation" on page 58

CONFigure:DPD:FNAMe <FileName>

This command defines a name for the DPD correction table.

Parameters:

<FileName> String containing the DPD table file name.

Example: `CONF:DPD:FNAM 'DPDTable'`

Defines the table name 'DPDTable'.

Manual operation: See "Selecting the DPD shaping method" on page 58

CONFigure:DPD:SEQuence <State>

This command selects the order in which the AM/AM and AM/PM distortion are calculated.

Available when both `CONFigure:DPD:AMAM[:STATe]` and `CONFigure:DPD:AMPM[:STATe]` have been turned on.

Parameters:

<Order>

AMFirst
Calculates the AM/AM distortion first, then the AM/PM distortion.

PMFirst
Calculates the AM/PM distortion first, then the AM/AM distortion.

Example: `CONF:DPD:SEQ AMF`
Calculates the AM/AM curve first.

Manual operation: See "Selecting the order of model calculation" on page 58

CONFigure:DPD:SHAPing:MODE <Method>

This command selects the method use to shape the DPD function.

Parameters:

<Method>

POLYnomial
DPD function based on the characterstics of the polynomial system model.

TABLe
DPD function based on the correction values kept in a table calculated by the R&S SMW.

*RST: TABLe

Example: `CONF:DPD:SHAP:MODE TABL`
DPD function based on correction values kept in a table.

Manual operation: See "Selecting the DPD shaping method" on page 58

CONFigure:DPD:TRADeoff <Percentage>

This command defines the power / linearity tradoff for DPD calculation.

Parameters:

<Percentage> <numeric value>

Default unit: PCT

Example: `CONF:DPD:TRAD 75`
Defines a tradeoff of 75 %.

Manual operation: See "DPD Power / Linearity Tradeoff" on page 59

CONFigure:DPD:UPDate

This command updates the DPD shaping tables on the R&S SMW when modeling parameters have changed.

Example: `CONF:DPD:UPD`
Updates the shaping table.

Usage: Event

CONFigure:DPD:UPDate:LEDState?

This command queries the state of an update of the shaping table.

Return values:

<State> **GREen**
Transmission was successful.
GREY
Unknown transmission state.
RED
Transmission was not successful.

Example: `CONF:DPD:UPD`
`CONF:DPD:UPD:LEDS?`
would return, e.g.:
`GREY`

Usage: Query only

5.6.12 Configuring Envelope Tracking

CONFigure:PAE:ICHannel:MULTiplier <Multiplier>

This command defines a multiplier to take into account various effects resulting from the measurement equipment connected to the I channel.

Parameters:

<Multiplier> <numeric value>

Example: `CONF:PAE:ICH:MULT 0.75`
Defines a multiplier of 0.75.

CONFigure:PAE:ICHannel:OFFSet <Offset>

This command defines an offset for the I channel.

Parameters:

<Offset> <numeric value>
Default unit: No unit

Example: `CONF:PAE:ICH:EOFF 1`
Defines an offset of 1.

CONFigure:PAE:ICHannel:RESistor <Resistance>

This command defines the characteristics of the shunt resistor used in the test setup.

Parameters:

<Resistance> <numeric value>

Resistance in Ohm.

Example: `CONF:PAE:ICH:RES 1.5`

Defines a resistance of 1.5 Ohm.

CONFigure:PAE:QCHannel:MULTiplier <Multiplier>

This command defines a multiplier to take into account various effects resulting from the measurement equipment connected to the Q channel.

Parameters:

<Multiplier> <numeric value>

Example: `CONF:PAE:QCH:MULT 1.2`

Defines a multiplier of 1.2.

CONFigure:PAE:QCHannel:OFFSet <Offset>

This command defines an offset for the Q channel.

Parameters:

<Offset> <numeric value>

Default unit: No unit

Example: `CONF:PAE:QCH:OFFS 1`

Defines an offset of 1.

5.6.13 Configuring ACP Measurements

CALCulate<n>:MARKer<m>:FUNCtion:POWer:RESult? <Measurement>

This command queries the (numerical) results of the ACP measurement.

Suffix:

<n> irrelevant

<m> irrelevant

Query parameters:

<Measurement> **ACP**
Queries the results of the ACP measurement.
Returns the power for every active transmission and adjacent channel. The order is:
• power of the transmission channels
• power of adjacent channel (lower,upper)

Example: `CALC:MARK:FUNC:POW:RES?`
would return, e.g.
`-21.76, 3.21, 2.57`

Usage: Query only

Manual operation: See "Adjacent Channel Power (ACP)" on page 13

[SENSe:]POWer:ACHannel:AABW <State>

This command turns automatic selection of the measurement bandwidth for ACP measurements on and off.

When you turn this on, the application selects a measurement bandwidth that is large enough to capture all channels evaluated by the ACP measurement.

Parameters:

<State> ON | OFF

Example: `POW:ACH:AABW ON`
Turns on automatic selection of the measurement bandwidth.

Manual operation: See "Selecting the measurement bandwidth" on page 63

[SENSe:]POWer:ACHannel:ACPairs? <ChannelNumber>

This command selects the number of adjacent channels evaluated in the ACP measurement.

Parameters:

<ChannelNumber> <numeric value>: (integer only)
Number of adjacent channels to the left and right of the Tx channel.
*RST: 1

Example: `POW:ACH:ACP 2`
Evaluates two pairs of adjacent channels in the measurement.

Usage: Query only

Manual operation: See "Defining characteristics of the adjacent channels" on page 64

[SENSe:]POWer:ACHannel:BANDwidth:ACHannel <Bandwidth>

This command defines the bandwidth of the adjacent channels evaluated in the ACP measurement.

Parameters:

<Bandwidth> <numeric value>

Default unit: Hz

Example: `POW:ACH:BAND:ACH 5MHZ`
Defines a bandwidth of 5 MHz for all adjacent channels.

Manual operation: See "Defining characteristics of the adjacent channels" on page 64

[SENSe:]POWer:ACHannel:BANDwidth[:CHANnel] <Bandwidth>

This command defines the bandwidth of the transmission channel evaluated in the ACP measurement.

Parameters:

<Bandwidth> <numeric value>

Default unit: Hz

Example: `POW:ACH:BAND?`
Queries the bandwidth of the transmission channel.

Manual operation: See "Defining characteristics of the transmission channel" on page 64

[SENSe:]POWer:ACHannel:SPACing:ACHannel <Bandwidth>

This command defines the channel spacing in ACP measurement.

Parameters:

<Bandwidth> <numeric value>

Distance between the center frequency of one channel and the center frequency of the next channel.

Default unit: Hz

Example: `POW:ACH:SPAC:ACH 10MHZ`
Defines a channel spacing of 10 MHz.

Manual operation: See "Defining characteristics of the adjacent channels" on page 64

5.6.14 Configuring Power Measurements

CONFigure:POWer:RESult:P3DB:REFerence <InputPower>

This command defines the input power corresponding to the gain reference required to calculate the Compression Points.

The command is available when you turn `CONFigure:POWer:RESult:P3DB[:STATe]` off.

Parameters:

<InputPower> <numeric value>

Default unit: dBm

Example:
`CONF:POW:RES:P3DB OFF`
`CONF:POW:RES:P3DB:REF 3`
Reference point is the gain measured at a input power of 3 dBm.

Manual operation: See "Configuring compression point calculation" on page 62

CONFigure:POWer:RESult:P3DB[:STATe] <State>

This command turns automatic calculation of the reference point required to determine the Compression Points (1 dB, 2 dB and 3 dB) on and off.

Parameters:

<State> ON | OFF

*RST: ON

Example:
`CONF:POW:RES:P3DB ON`
Automatically determines the reference point.

Manual operation: See "Configuring compression point calculation" on page 62

CONFigure:POWer:RESult:PONLy[:STATe] <State>

This command turns evaluation of all results (numerical and graphical) except the output power on and off.

Parameters:

<State>

ON
Only the output power of the amplifier is calculated.

OFF
All results are evaluated.

*RST: OFF

Example:
`CONF:POW:RES:PONL ON`
Evaluates the output power only.

Manual operation: See "Evaluating only the DUT output power" on page 62

5.6.15 Configuring Parameter Sweeps

CONFigure:PSWeep:ADJust:LEVel[:STATe] <State>

This command turns synchronization of the generator output level and the analyzer reference level on and off.

The command is available when one of the parameters used in the Parameter Sweep is the "Generator Power".

When you synchronize the levels, it is recommended to also define the expected gain of the DUT with `CONFigure:PSWeep:EXPected:GAIN`.

Parameters:
<State> ON | OFF

Example: `CONF:PSW:ADJ:LEV ON`
Synchronizes the generator output level and the analyzer reference level.

Manual operation: See "Synchronizing the levels of signal generator and analyzer" on page 67

CONFigure:PSWeep:EXPected:GAIN <Gain>

This command defines the expected gain of the DUT.

This is necessary when you synchronize the generator output level and the reference level of the analyzer `CONFigure:PSWeep:ADJust:LEVel[:STATe]` = ON.

The command is available when one of the parameters used in the Parameter Sweep is the "Generator Power".

Parameters:
<Gain> <numeric value>

Default unit: dB

Example:
```
CONF:PSW:ADJ:LEV ON
CONF:PSW:EXP:GAIN 5
```
Defines an expected gain of 5 dB.

Manual operation: See "Synchronizing the levels of signal generator and analyzer" on page 67

CONFigure:PSWeep[:STATe] <State>

This command turns the Parameter Sweep on and off.

Parameters:

<State> ON | OFF

Example: `CONF:PSW ON`
Turns on the Parameter Sweep.

Manual operation: See "Turning the parameter sweep on and off" on page 66

CONFigure:PSWeep:X:SETTing <Setting>

This command selects the parameter type for the first parameter controlled by the Parameter Sweep.

Parameters:

<Setting>

BIAS
Controls the envelope bias.

DELay
Controls the delay between envelope and RF signal.

FREQuency
Controls the frequency.

POWer
Controls the output level.

Example: See `CONFigure:PSWeep:Y:SETTing`.

CONFigure:PSWeep:X:STARt <Start>

This command defines the start value for the first parameter controlled by the Parameter Sweep.

Parameters:

<Start> <numeric value> whose unit depends on the parameter type you have selected with `CONFigure:PSWeep:Y:SETTing`:

- Hz in case of the center frequency
- dBm in case of the output level
- s in case of the delay between envelope and RF signal
- V in case of the envelope bias

Default unit: UNITS_PS

Example: See `CONFigure:PSWeep:Y:SETTing`.

CONFigure:PSWeep:X:STEP <StepSize>

This command defines the stepsize for the first parameter controlled by the Parameter Sweep.

Parameters:

<StepSize> <numeric value> whose unit depends on the parameter type you have selected with `CONFigure:PSWeep:Y:SETTing`:
• Hz in case of the center frequency
• dBm in case of the output level
• s in case of the delay between envelope and RF signal
• V in case of the envelope bias

Default unit: UNITS_PS_STEP

Example: See `CONFigure:PSWeep:Y:SETTing`.

CONFigure:PSWeep:X:STOP <Stop>

This command defines the stop value for the first parameter controlled by the Parameter Sweep.

Parameters:

<Stop> <numeric value> whose unit depends on the parameter type you have selected with `CONFigure:PSWeep:Y:SETTing`:
• Hz in case of the center frequency
• dBm in case of the output level
• s in case of the delay between envelope and RF signal
• V in case of the envelope bias

Default unit: UNITS_PS

Example: See `CONFigure:PSWeep:Y:SETTing`.

CONFigure:PSWeep:Y:SETTing <Setting>

This command selects the parameter type for the second parameter controlled by the Parameter Sweep.

Parameters:

<Setting>

BIAS
Controls the envelope bias.

DELay
Controls the delay between envelope and RF signal.

FREQuency
Controls the frequency.

POWer
Controls the output level.

Example:

```
CONF:PSW:Y:STAT ON
CONF:PSW:Y:SETT FREQ
CONF:PSW:Y:STAR 10MHZ
CONF:PSW:Y:STOP 100MHZ
CONF:PSW:Y:STEP 1MHZ
```

Configure the second parameter with start, stop and stepsize values.

CONFigure:PSWeep:Y:STARt <Start>

This command defines the start value for the second parameter controlled by the Parameter Sweep.

Parameters:

<Start>

<numeric value> whose unit depends on the parameter type you have selected with `CONFigure:PSWeep:Y:SETTing`:
- Hz in case of the center frequency
- dBm in case of the output level
- s in case of the delay between envelope and RF signal
- V in case of the envelope bias

Default unit: UNITS_PS

Example: See `CONFigure:PSWeep:Y:SETTing`.

CONFigure:PSWeep:Y:STATe <State>

This command turns the second parameter controlled by the Parameter Sweep on and off.

Parameters:

<State>

ON | OFF

*RST: ON

Example: See `CONFigure:PSWeep:Y:SETTing`.

CONFigure:PSWeep:Y:STEP <StepSize>

This command defines the stepsize for the second parameter controlled by the Parameter Sweep.

Parameters:

<StepSize>

<numeric value> whose unit depends on the parameter type you have selected with `CONFigure:PSWeep:Y:SETTing`:
- Hz in case of the center frequency
- dBm in case of the output level
- s in case of the delay between envelope and RF signal
- V in case of the envelope bias

Default unit: UNITS_PS_STEP

Example: See `CONFigure:PSWeep:Y:SETTing`.

CONFigure:PSWeep:Y:STOP <Stop>

This command defines the stop value for the second parameter controlled by the Parameter Sweep.

Parameters:

<Stop> <numeric value> whose unit depends on the parameter type you have selected with `CONFigure:PSWeep:Y:SETTing`:
• Hz in case of the center frequency
• dBm in case of the output level
• s in case of the delay between envelope and RF signal
• V in case of the envelope bias
Default unit: UNITS_PS

Example: See `CONFigure:PSWeep:Y:SETTing`.

5.7 Analyzing Results

5.7.1 Configuring Traces

DISPlay[:WINDow<n>]:TRACe<t>:MODE <Trace>

This command selects the traces to be displayed in the graphical result displays.

Suffix:

<n> 1..n

<t> 1..n

Parameters:

<Trace> Available traces depend on the result display. See table below for details.

BBI
Selects the trace showing data recorded on the baseband I channel.

BLANk
Turns the trace off.

BBPower
Selects the trace showing the combined data of the I and Q channel.

BBQ
Selects the trace showing data recorded on the baseband Q channel.

MODel
Selects the trace showing the modeled signal.

REFerence
Selects the trace showing the reference signal.

RF
Selects the trace showing the measured signal recorded on the RF input.

WRIT
Selects the clear write trace.

Example: e.g. for the AM/AM result display:

```
DISP:WIND:TRAC1:MODE RF
DISP:WIND:TRAC2:MODE MOD
```

Displays the measured and the modeled signal.
e.g. for the Spectrum EVM result display:

```
DISP:WIND:TRAC1:MODE RF
DISP:WIND:TRAC2:MODE BLAN
```

Displays the measured signal and hides the modeled signal.

Result display	Supported traces
AM/AM	RF (always trace 1) MODel (always trace 2) BLANk (for both traces)
AM/PM	RF (always trace 1) MODel (always trace 2) BLANk (for both traces)
Gain Compression	RF (always trace 1)
Magnitude Capture (RF, I and Q)	WRITe (always trace 1)
PAE Input Power	WRITe (always trace 1)
PAE Time	WRITe (always trace 1)
Power vs Time	WRITe (always trace 1)

Result display	Supported traces
Raw EVM	REFerence (always trace 1) MODel (always trace 2) BLANk (for both traces)
Spectrum EVM	REFerence (always trace 1) MODel (always trace 2) BLANk (for both traces)
Spectrum FFT RF	RF (always trace 1) MODel (always trace 2) REFerence (always trace 3) BLANk (for all three traces)
Spectrum FFT I	WRITe (always trace 1)
Spectrum FFT Q	WRITe (always trace 1)
Time Domain	RF (always trace 1) MODel (always trace 2) REFerence (always trace 3) BBI (always trace 4) BBQ (always trace 5) BBP (always trace 6) BLANk (for all six traces)

5.7.2 Using Markers

5.7.2.1 General Marker Settings

DISPlay:MTABle <DisplayMode>

This command turns the marker table on and off.

Parameters:
<DisplayMode>

ON
Turns the marker table on.

OFF
Turns the marker table off.

AUTO
Turns the marker table on if 3 or more markers are active.

*RST: AUTO

Example: `DISP:MTAB ON`
Activates the marker table.

Manual operation: See "Marker Table Display" on page 72

5.7.2.2 Configuring Individual Markers

CALCulate<n>:DELTamarker<m>:AOFF

This command turns *all* delta markers off.

(<m> is irrelevant)

Example: `CALC:DELT:AOFF`
Turns all delta markers off.

Usage: Event

CALCulate<n>:DELTamarker<m>:MREF <Reference>

This command selects a reference marker for a delta marker other than marker 1.

Parameters:
<Reference>

Example: `CALC:DELT3:MREF 2`
Specifies that the values of delta marker 3 are relative to marker 2.

Manual operation: See "Reference Marker" on page 72

CALCulate<n>:DELTamarker<m>[:STATe] <State>

This command turns delta markers on and off.

If necessary, the command activates the delta marker first.

No suffix at DELTamarker turns on delta marker 1.

Parameters:

<State> ON | OFF

*RST: OFF

Example: `CALC:DELT2 ON`
Turns on delta marker 2.

Manual operation: See "Marker State" on page 71
See "Marker Type" on page 71

CALCulate<n>:DELTamarker<m>:TRACe <Trace>

This command selects the trace a delta marker is positioned on.

Note that the corresponding trace must have a trace mode other than "Blank".

If necessary, the command activates the marker first.

Parameters:

<Trace> Trace number the marker is assigned to.

Example: `CALC:DELT2:TRAC 2`
Positions delta marker 2 on trace 2.

CALCulate<n>:DELTamarker<m>:X <Position>

This command moves a delta marker to a particular coordinate on the x-axis.

If necessary, the command activates the delta marker and positions a reference marker to the peak power.

Parameters:

<Position> Numeric value that defines the marker position on the x-axis.

Range: The value range and unit depend on the measurement and scale of the x-axis.

Example: `CALC:DELT:X?`
Outputs the absolute x-value of delta marker 1.

Manual operation: See "Marker Position (X-value)" on page 71

CALCulate<n>:DELTamarker<m>:Y?

This command queries the relative position of a delta marker on the y-axis.

If necessary, the command activates the delta marker first.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single measurement mode.

See also `INITiate<n>:CONTinuous` on page 99.

The unit depends on the application of the command.

Return values:

<Position> Position of the delta marker in relation to the reference marker or the fixed reference.

Example:
`INIT:CONT OFF`
Switches to single sweep mode.
`INIT;*WAI`
Starts a sweep and waits for its end.
`CALC:DELT2 ON`
Switches on delta marker 2.
`CALC:DELT2:Y?`
Outputs measurement value of delta marker 2.

Usage: Query only

CALCulate<n>:MARKer<m>:AOFF

This command turns all markers off.

Example:
`CALC:MARK:AOFF`
Switches off all markers.

Usage: Event

Manual operation: See "All Markers Off" on page 72

CALCulate<n>:MARKer<m>[:STATe] <State>

This command turns markers on and off. If the corresponding marker number is currently active as a deltamarker, it is turned into a normal marker.

Parameters:

<State> ON | OFF

*RST: OFF

Example:
`CALC:MARK3 ON`
Switches on marker 3.

Manual operation: See "Marker State" on page 71
See "Marker Type" on page 71

CALCulate<n>:MARKer<m>:TRACe <Trace>

This command selects the trace the marker is positioned on.

Note that the corresponding trace must have a trace mode other than "Blank".

If necessary, the command activates the marker first.

Parameters:
<Trace> **1 to 4**
Trace number the marker is assigned to.

Example: `CALC:MARK3:TRAC 2`
Assigns marker 3 to trace 2.

Manual operation: See "Assigning the Marker to a Trace" on page 72

CALCulate<n>:MARKer<m>:X <Position>

This command moves a marker to a particular coordinate on the x-axis.

If necessary, the command activates the marker.

If the marker has been used as a delta marker, the command turns it into a normal marker.

Parameters:
<Position> Numeric value that defines the marker position on the x-axis.
The unit depends on the result display.
Range: The range depends on the current x-axis range.

Example: `CALC:MARK2:X 1.7MHz`
Positions marker 2 to frequency 1.7 MHz.

Manual operation: See "Marker Position (X-value)" on page 71

CALCulate<n>:MARKer<m>:Y?

This command queries the position of a marker on the y-axis.

If necessary, the command activates the marker first.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single measurement mode.

See also `INITiate<n>:CONTinuous` on page 99.

Return values:
<Result> Result at the marker position.

Example: `INIT:CONT OFF`
Switches to single measurement mode.
`CALC:MARK2 ON`
Switches marker 2.
`INIT;*WAI`
Starts a measurement and waits for the end.
`CALC:MARK2:Y?`
Outputs the measured value of marker 2.

Usage: Query only

5.7.2.3 Positioning Markers

CALCulate<n>:DELTamarker<m>:MAXimum:LEFT

This command moves a delta marker to the next higher value.

The search includes only measurement values to the left of the current marker position.

Usage: Event

CALCulate<n>:DELTamarker<m>:MAXimum:NEXT

This command moves a marker to the next higher value.

Usage: Event

Manual operation: See "Search Next Peak" on page 73

CALCulate<n>:DELTamarker<m>:MAXimum[:PEAK]

This command moves a delta marker to the highest level.

If the marker is not yet active, the command first activates the marker.

Usage: Event

Manual operation: See "Peak Search" on page 73

CALCulate<n>:DELTamarker<m>:MAXimum:RIGHt

This command moves a delta marker to the next higher value.

The search includes only measurement values to the right of the current marker position.

Usage: Event

CALCulate<n>:DELTamarker<m>:MINimum:LEFT

This command moves a delta marker to the next higher minimum value.

The search includes only measurement values to the right of the current marker position.

Usage: Event

CALCulate<n>:DELTamarker<m>:MINimum:NEXT

This command moves a marker to the next higher minimum value.

Usage: Event

Manual operation: See "Search Next Minimum" on page 73

CALCulate<n>:DELTamarker<m>:MINimum[:PEAK]

This command moves a delta marker to the minimum level.

If the marker is not yet active, the command first activates the marker.

Usage: Event

Manual operation: See "Search Minimum" on page 73

CALCulate<n>:DELTamarker<m>:MINimum:RIGHt

This command moves a delta marker to the next higher minimum value.

The search includes only measurement values to the right of the current marker position.

Usage: Event

CALCulate<n>:MARKer<m>:MAXimum:LEFT

This command moves a marker to the next lower peak.

The search includes only measurement values to the left of the current marker position.

Usage: Event

CALCulate<n>:MARKer<m>:MAXimum:NEXT

This command moves a marker to the next lower peak.

Usage: Event

Manual operation: See "Search Next Peak" on page 73

CALCulate<n>:MARKer<m>:MAXimum[:PEAK]

This command moves a marker to the highest level.

If the marker is not yet active, the command first activates the marker.

Usage: Event

Manual operation: See "Peak Search" on page 73

CALCulate<n>:MARKer<m>:MAXimum:RIGHt

This command moves a marker to the next lower peak.

The search includes only measurement values to the right of the current marker position.

Usage: Event

CALCulate<n>:MARKer<m>:MINimum:LEFT

This command moves a marker to the next minimum value.

The search includes only measurement values to the right of the current marker position.

Usage: Event

CALCulate<n>:MARKer<m>:MINimum:NEXT

This command moves a marker to the next minimum value.

Usage: Event

Manual operation: See "Search Next Minimum" on page 73

CALCulate<n>:MARKer<m>:MINimum[:PEAK]

This command moves a marker to the minimum level.

If the marker is not yet active, the command first activates the marker.

Usage: Event

Manual operation: See "Search Minimum" on page 73

CALCulate<n>:MARKer<m>:MINimum:RIGHt

This command moves a marker to the next minimum value.

The search includes only measurement values to the right of the current marker position.

Usage: Event

5.7.3 Configuring Numerical Result Displays

DISPlay[:WINDow<n>]:PTABle:ITEM <Item>, <State>
DISPlay[:WINDow<n>]:PTABle:ITEM? <Item>

This command adds and removes results from the Parameter Sweep Table.

Suffix:

<n> 1..n

Parameters:

<State> ON | OFF

*RST: All results are ON.

Parameters for setting and query:

<Item> Selects the result.
See the table at `CONFigure:PSWeep:Z<n>:RESult` for a list of available parameters.

Example:
`DISP:PTAB:ITEM RMS,OFF`
Removes the RMS Power result from the Parameter Sweep Table.

DISPlay[:WINDow<n>]:TABLe:ITEM <Item>, <State>
DISPlay[:WINDow<n>]:TABLe:ITEM? <Item>

This command adds and removes results from the Result Summary.

Suffix:

<n> 1..n
Note that you have to include the `WINDow` syntax element if the Result Summary is in a window other than window 1.

Parameters:

<State> ON | OFF

*RST: All results are ON.

Parameters for setting and query:

<Item> Selects the result.
See the table below for a list of available parameters.

Example:
`DISP:TABL:ITEM GIMB,OFF`
Removes the Gain Imbalance result from the Result Summary.
`DISP:WIND2:TABL:ITEM? APAE`
would return, e.g.
`1`

SCPI parameter	Result
AMW	AM Curve Width
APAE	Average PAE
BBIV	Baseband I Input Voltage
BBP	Baseband Power
BBQV	Baseband Q Input Voltage
CFIN	Crest Factor In
CFOU	Crest Factor Out
FERR	Frequency Error
GAIN	Gain
GIMB	Gain Imbalance
ICC	Current
IQIM	I/Q Imbalance
IQOF	I/Q Offset
MERR	Magnitude Error
P1DB	1 dB Compression Point
P2DB	2 dB Compression Point
P3DB	3 dB Compression Point
PERR	Phase Error
PINP	Power In
PMW	PM Curve Width
POUT	Power Out
QERR	Quadrature Error
REVM	Raw EVM
RMEV	Raw Model EVM
SRER	Sample Rate Error
VCC	Voltage

5.7.4 Configuring Result Display Characteristics

CALCulate<n>:GAIN:X <ResultType>

This command selects the type of information displayed on x-axis in the Gain Compression result display.

Suffix:
<n> 1..n

Parameters:
<ResultType> **PINPut**
Shows the gain compression against the input level.
POUTput
Shows the gain compression against the output level.

Example: `CALC:GAIN:X PINP`
Displays the gain against the input level.

Manual operation: See "Configuring the Gain Compression Result Display" on page 76

CALCulate<n>:UNIT:ANGLe <Unit>

This command selects the unit that the phase is shown in the AM/PM result display.

Suffix:
<n> 1..n

Parameters:
<Unit> **DEG**
Phase displayed in degrees.
RAD
Phase displayed in radians.

Example: `CALC:UNIT:ANGL DEG`
Shows the phase results in degrees.

Manual operation: See "Configuring the AM/PM result display" on page 76

CONFigure:PSWeep:Z<n>:RESult <Result>

This command selects the result type displayed on the y-axis of the paramater sweep diagram.

Suffix:
<n> 1..n

Parameters:
<Result> See table below for supported result types.

Example: `CONF:PSW:Z:RES EVM`
Displays the EVM against two parameters in the Parameter Sweep result display.

Manual operation: See "Selecting the result type displayed in the Parameter Sweep diagram" on page 76

ACL1	ACP Adjacent 1 Lower
ACP	Adjacent Channel Power
ACU1	ACP Adjacent 1 Upper
AMWidth	AM/AM Curve Width
CFACtor	Crest Factor
EVM	EVM
GAIN	Gain
ICC	Current (I_cc)
PAE	PAE
PMWidth	AM/PM Curve Width
RMS	RMS Power
VCC	Voltage (V_cc)
VICC	Power (V_cc * I_cc)

DISPlay[:WINDow<n>]:TDOMain:X[:SCALe]:DURation? <Time>

This command defines the amount of data displayed on the x-axis of the Time Domain result display.

The command is available when `DISPlay[:WINDow<n>]:TDOMain:X[:SCALe]:MODE` has been turned off.

Suffix:
<n> 1..n

Parameters:
<Time> <numeric value>

Time that is displayed on the x-axis, beginning at the offset defined with `DISPlay[:WINDow<n>]:TDOMain:X[:SCALe]:OFFSet?`.

Default unit: s

Example:
```
DISP:TDOM:X:MODE OFF
DISP:TDOM:X:DUR 12us
```
Scales the x-axis to display 12 µs in the Time Domain result display.

Usage: Query only

Manual operation: See "Configuring the Time Domain result display" on page 75

DISPlay[:WINDow<n>]:TDOMain:X[:SCALe]:MODE <State>

This command turns automatic scaling of the x-axis in the Time Domain result display on and off.

Suffix:

<n> 1..n

Parameters:

<State> **ON**
Turns on automatic scaling of the x-axis.
OFF
Turns on manual scaling of the x-axis.

Example: `DISP:TDOM:X:MODE OFF`
Turns on manual scaling of the x-axis.

Manual operation: See "Configuring the Time Domain result display" on page 75

DISPlay[:WINDow<n>]:TDOMain:X[:SCALe]:OFFSet? <Time>

This command defines the origin of the x-axis in the Time Domain result display.

The command is available when `DISPlay[:WINDow<n>]:TDOMain:X[:SCALe]:MODE` has been turned off.

Suffix:

<n> 1..n

Parameters:

<Time> <numeric value>
Time offset relative to the first recorded sample (when synchronization has failed) or the first sample of the synchronized data (when synchronization was successful).
Default unit: s

Example: `DISP:TDOM:X:MODE OFF`
`DISP:TDOM:X:OFFS 12us`
Defines an offset of 12 µs.

Usage: Query only

Manual operation: See "Configuring the Time Domain result display" on page 75

DISPlay[:WINDow<n>]:TDOMain:Y[:SCALe]:NORMalise[:STATe] <State>

This command turns normalization of the results in the Time Domain result display on and off.

Suffix:

<n> 1..n

Parameters:

<State> ON | OFF

Example: `DISP:TDOM:Y:NORM ON`
Normalizes the results in the Time Domain result display to 1.

Manual operation: See "Configuring the Time Domain result display" on page 75

5.7.5 Scaling the Diagram Axes

DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:AUTO <State>

This command turns automatic scaling of the x-axis in graphical result displays on and off.

Suffix:

<n> 1..n

<t> 1..n

Parameters:

<State>

OFF
Selects manual scaling of the diagram.

ON
Automatically scales the diagram when new results are available.

ONCE
Automaically scales the diagram once whenever required.

Example: `DISP:TRAC:X:AUTO ON`
Scales the axis each time new results are available.

Manual operation: See "Scaling the x-axis automatically" on page 77

DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:MAXimum <Value>

This command defines the value at the top of the x-axis.

Suffix:

<n> 1..n

<t> 1..n

Parameters:

<Value> <numeric value>

Default unit: Depends on the result display.

Example:
```
DISP:TRAC:x:AUTO OFF
DISP:TRAC:x:MIN -10DBM
DISP:TRAC:x:MAX -110DBM
```
The x-axis covers a level range of 100 dB.

Manual operation: See "Scaling the x-axis manually" on page 78

DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:MINimum <Value>

This command defines the value at the bottom of the y-axis.

Suffix:

<n> 1..n

<t> 1..n

Parameters:

<Value> <numeric value>

Default unit: Depends on the result display.

Example:
```
DISP:TRAC:X:AUTO OFF
DISP:TRAC:X:MIN -10DBM
DISP:TRAC:X:MAX -110DBM
```
The x-axis covers a level range of 100 dB.

Manual operation: See "Scaling the x-axis manually" on page 78

DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:PDIVision <Distance>

This command defines the distance between the horizontal grid lines in graphical result displays.

Available when you turn off automatic scaling with `DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:AUTO`.

Suffix:

<n> 1..n

<t> 1..n

Parameters:

<Distance> <numeric value>

Default unit: Depends on the result display.

Example:
```
DISP:TRAC:X:SCAL:AUTO OFF
DISP:TRAC:X:PDIV 5DBM
```
Defines a distance of 5 dBm between the grid lines.

Manual operation: See "Scaling the x-axis manually" on page 78

DISPlay[:WINDow<n>]:TRACe<t>:X[:SCALe]:UNIT?

This command queries the unit of the x-axis

Suffix:

<n> 1..n
Selects the measurement window.

<t> irrelevant

Return values:

<Unit> Unit of the x-axis in the selected window.

Example: `DISP:WIND4:TRAC:X:UNIT?`
would return, e.g.
`SEC`

Usage: Query only

DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:AUTO <State>

This command turns automatic scaling of the y-axis in graphical result displays on and off.

Suffix:

<n> 1..n

<t> 1..n

Parameters:

<State> **OFF**
Selects manual scaling of the diagram.
ON
Automatically scales the diagram when new results are available.
ONCE
Automaically scales the diagram once whenever required.

Example: `DISP:TRAC:Y:AUTO ON`
Scales the axis each time new results are available.

Manual operation: See "Scaling the y-axis automatically" on page 79

DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:MAXimum <Value>

This command defines the value at the top of the y-axis.

Suffix:

<n> 1..n

<t> 1..n

Parameters:

<Value> <numeric value>
Default unit: Depends on the result display.

Example:
```
DISP:TRAC:Y:AUTO OFF
DISP:TRAC:Y:MIN -10DBM
DISP:TRAC:Y:MAX -110DBM
```
The y-axis covers a level range of 100 dB.

Manual operation: See "Scaling the y-axis manually" on page 79

DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:MINimum <Value>

This command defines the value at the bottom of the y-axis.

Suffix:

<n> 1..n

<t> 1..n

Parameters:

<Value> <numeric value>

Default unit: Depends on the result display.

Example:
```
DISP:TRAC:Y:AUTO OFF
DISP:TRAC:Y:MIN -10DBM
DISP:TRAC:Y:MAX -110DBM
```
The y-axis covers a level range of 100 dB.

Manual operation: See "Scaling the y-axis manually" on page 79

DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:PDIVision <Distance>

This command defines the distance between the grid lines in graphical result displays.

Available when you turn off automatic scaling with `DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:AUTO`.

Suffix:

<n> 1..n

<t> 1..n

Parameters:

<Distance> <numeric value>

Default unit: Depends on the result display.

Example:
```
DISP:TRAC:Y:SCAL:AUTO OFF
DISP:TRAC:Y:PDIV 5DBM
```
Defines a distance of 5 dBm between the grid lines.

Manual operation: See "Scaling the y-axis manually" on page 79

DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:RPOSition <Position>

This command defines the position of the reference value.

You can define the reference value with `DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:RVALue`.

Suffix:

<n>	1..n
<t>	1..n

Parameters:

<Position>	<numeric value> Default unit: %

Example:

```
DISP:TRAC:Y:AUTO OFF
DISP:TRAC:Y:RVAL 0DBM
DISP:TRAC:Y:RPOS 80
```

Positions the reference value at the 80 % mark of the y-axis.

Manual operation: See "Scaling the y-axis manually" on page 79

DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:RVALue <Reference>

This command defines the reference value of a result display.

Available when you turn off automatic scaling with `DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:AUTO`.

Suffix:

<n>	1..n
<t>	1..n

Parameters:

<Reference>	<numeric value> Default unit: The unit depends on the result display.

Example:

```
DISP:TRAC:Y:AUTO OFF
DISP:TRAC:Y:RVAL 10DB
```

Defines a reference value of a 10 dB.

Manual operation: See "Scaling the y-axis manually" on page 79

DISPlay[:WINDow<n>]:TRACe<t>:Y[:SCALe]:UNIT?

This command queries the unit of the y-axis

Suffix:

<n>	1..n Selects the measurement window.
<t>	irrelevant

Return values:

<Unit>	Unit of the y-axis in the selected window.

Example: `DISP:WIND3:TRAC:Y:UNIT?`
would return, e.g.
`DBM`

Usage: Query only

5.7.6 Managing Measurement Data

MMEMory:LOAD:IQ:STATe <1>, <FileName>

This command stores the currently captured I/Q data to a file.

After restoring the I/Q data, the application also analyzes the data again.

Setting parameters:
<1>

<FileName> String containing the path and file name.

Example: `MMEM:LOAD:IQ:STAT 1,'C:\IQData\Amplfier.iq.tar'`
Restores the specified I/Q data.

Usage: Setting only

MMEMory:STORe<n>:IQ:COMMent <Comment>

This command defines a comment for I/Q data you want to store.

Suffix:
<n> irrelevant

Setting parameters:
<Comment> String containing the comment.

Example: See `MMEMory:STORe<n>:IQ:STATe`.

MMEMory:STORe<n>:IQ:STATe <Number>, <FileName>

This command stores the currently captured I/Q data to a file.

In secure user mode, settings that are to be stored on the instrument are stored to volatile memory, which is restricted to 256 MB. Thus, a "Memory full" error may occur although the hard disk indicates that storage space is still available.

To store data permanently, select an external storage location such as a USB memory device.

For details see "Protecting Data Using the Secure User Mode" in the "Data Management" section of the R&S FSW User Manual.

Suffix:
<n> 1..n
irrelevant

Setting parameters:
<Number> Always '1'.

<FileName> String containing the path and file name.
The file type is .iq.tar.

Example:
```
MMEM:STOR:IQ:COMM 'A sensible comment'
MMEM:STOR:IQ:STAT 1,'C:\IQData\Amplfier.iq.tar'
```
Saves the I/Q data to the specified file and adds a sensible comment.

Usage: Setting only

MMEMory:STORe<n>:TRACe <Trace>, <FileName>

This command stores current trace data in a file.

Suffix:
<n> 1..n
irrelevant

Setting parameters:
<Trace> Number of the trace you want to save.
Note that the available number of traces depends on the selected result display.
Range: 1 to 6

<FileName> String containing the path and file name.

Example:
```
MMEM:STOR:TRAC 2,'C:\AmplifierTrace'
```
Saves the second trace in the specified directory.

Usage: Setting only

5.8 Deprecated Remote Commands for Amplifier Measurements

Following is a list of deprecated remote commands. The remote commands are still supported to maintain compatibility to previous versions of amplifier measurements, but it is strongly recommended to use the command system in the way it is meant to be used in the latest version of the R&S FSW-K18.

Legacy command	Replaced by	Comment
CONFigure:DPD:MODorder		
CONFigure:MODeling:AMAM:MORDer	CONFigure:MODeling:AMAM:ORDer	

Legacy command	Replaced by	Comment
CONFigure:MODeling:AMPM:MORDer	CONFigure:MODeling:AMPM:ORDer	
CONFigure:MODeling:ORDer	CONFigure:MODeling:SEQuence	
FETCh:POWer:CURRent[:RESult]	FETCh:BBPower:CURRent[:RESult]	
FETCh:POWer:MAXimum[:RESult]	FETCh:BBPower:MAXimum[:RESult]	
FETCh:POWer:MINimum[:RESult]	FETCh:BBPower:MINimum[:RESult]	

List of Commands

Index

F

G

H

I

K

L

M

N

O

P

R